DEVELOPMENT OF SOFTWARE FOR THE ANALYSIS AND DESIGN OF BUILDING FRAMES

Shaji M Jamal*

Jeevan Jacob**

Abstract:

This work deals with the development of a computer application for the analysis of structural frameworks including dynamic analysis and design of structural components. The project also aims to emphasize the importance of computers in the solution of everyday engineering problems, to understand the internal working and to know how the input data is being used by the computer in structural analysis and design programs. The developed program contains an analysis module which is used for the analysis of plane frames. It includes a module for the dynamic analysis of plane frames to find out the natural frequency and mode shapes of vibration. Other modules are for the design of reinforced concrete beams, columns and slabs.

* Asst. Prof., Dept of Civil Engg., Ilahia College of Engg & Tech., Muvattupuzha, Kerala.

** Asst.Prof., Dept of Civil Engg, M.A College of Engg., Kothamangalam, Kerala

INTRODUCTION

The introduction of computer and software in civil engineering industry has greatly reduced the complexities of different aspects in the analysis and design of projects, as well as reducing the amount of time necessary to complete the designs. Concurrently, this leads to greater savings in costs. More complex projects that were almost impossible to work out several years ago are now easily solved with the use of computers. But most of the standard structural analysis programs appear as *black boxes* to their users with input going in at one end and solution magically appears at the other end [1]. This study is mainly concerned with the structural theory and design procedures behind the structural programs and also the different steps involved in the development of a computer program for the analysis and design of structures.

OBJECTIVES

The main objectives of the study are:

- Develop a computer software for the analysis and design of structural elements
- To develop an in-depth appreciation of theoretical concepts.
- To learn the process of systematically developing engineering softwares.

SOFTWARE REVIEW

Software for the analysis and design of building frames was developed during the study. The following sections discuss the different components and capabilities of the software.

Design Module

The design Module includes sub-modules for design of column, beams and slabs. The design is based on IS: 456 – 2000 [2] and the method of design adopted is Limit State Method. Individual modules can be invoked from "Design" menu on the menu bar. Screen shot of the interface is shown in figure 1.

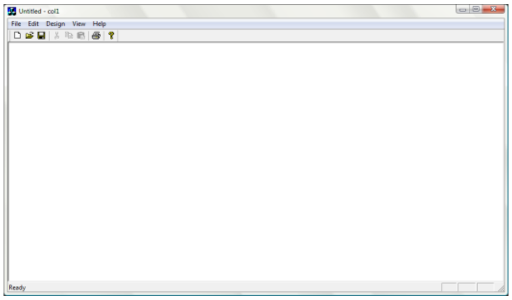

Figure 1. Design Module Main Window

Column Design Module

The Column Design sub-module can be invoked by using ‘Design□Column’ menu item on the menu bar of the main window. It can also be started using the shortcut key ‘F3’ from the main window. This module has only one dialog box (window). The dialog contains a number of controls like text boxes, labels, lists, etc. Interaction diagram similar to the charts in SP 16: 1980[3], is also integrated into the window for easy interpretation of results. Screen shot of the module is shown in figure 2. Data to be input for column design are:-

1. Loads
2. Member Properties
3. Reinforcement Details
4. Material Properties

The selection of material properties is made easy by the use of option buttons and the most commonly used grades are selected by default.

To design a column enter the data correctly and press the “Calculate” button. The module will start iterating and the result will be shown in the results area. If the column is unsafe, section properties, material properties (concrete) or bar diameter is to be changed and recalculated. Using “Mux-Pu Curve” or “Muy-Pu Curve” buttons, the Interaction diagrams can be displayed. Data for the Interaction diagram in both cases are also displayed on the window.

The results area shows the following:

Section load carrying capacity

Section moment capacity in x – direction

Section moment capacity in y – direction

Value of α_n which is calculated as per IS: 456–2000 (Cl. 39.6) [2]

Value of $\{(M_{ux}/M_{ux1})^{\alpha n} + (M_{uy}/M_{uy1})^{\alpha n}\}$

Percentage of steel

Area of steel and

The spacing of the two legged stirrups

Screen shot of the Column Design dialog box is shown in figure 2.

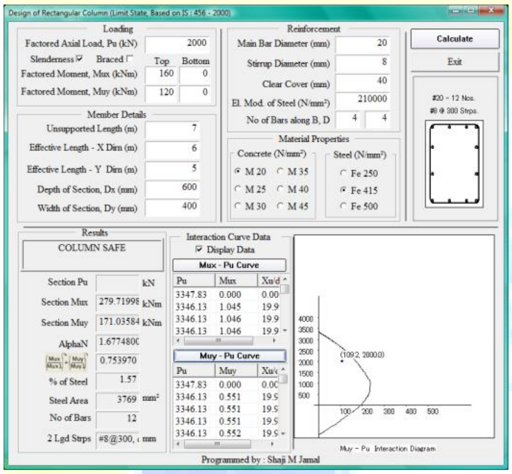

Figure 2. Column Design Window

The column design module can design both short column and slender columns.

Beam Design Module

This module also consists of only one window and it can be started from the main window using the 'Design□Beam' menu item from the main menu. The shortcut key 'F4' is assigned for invoking the beam design module in the main frame for easy access. The module designs beams based on IS: 456–2000[2] using Limit State Method. Design for shear and torsion are also incorporated into the design engine for both singly and doubly reinforced beams.

The screen shot of the module is shown in figure 3. The data to be supplied to the module are:

1. Loading
2. Member Details
3. Reinforcement Details
4. Material Properties

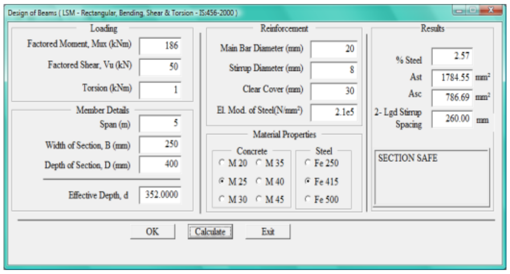

Figure 3. Beam Design Window

The design can be carried out by entering data correctly into the specified fields and then by pressing 'Calculate' button on the bottom of the dialog box. The results area shows the reinforcement details. The results we obtain from beam design module are the following:

- Percentage of steel, p_t
- Area of tension steel, A_{st}
- Area of compression steel, A_{sc}
- Two legged stirrup spacing, s_v

Slab Design Module

The slab design module consists of one window, the snapshot of which is shown in figure 4.

The input parameters are:

- Effective span in the longer direction
- Effective span in shorter direction
- No of discontinuous long edges
- No of discontinuous short edges
- Live load
- Basic span – depth ratio
- Overall depth
- Bar diameter
- Clear cover and
- Elastic modulus

While designing, if the aspect ratio is greater than 2, the module asks to design it as one way slab for which a second dialog will be opened.

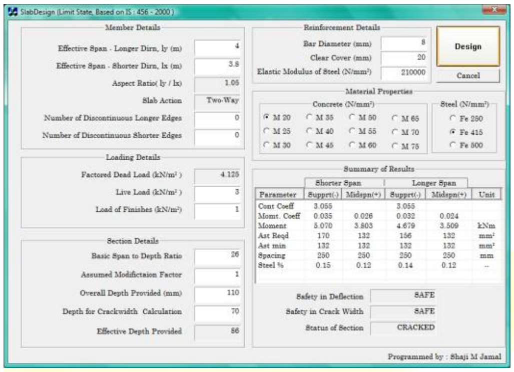

Figure 4. Slab Design Module

Analysis Module

The analysis module is used for the static and dynamic analysis of plane frames. The design of the analysis module window is done in such a way that all the necessary controls are readily available when the program is started to avoid confusion and time wastage. The snapshot of the analysis module is shown in figure 5. The white portion is the drawing area where the model as well as the post process is displayed. On extreme right end, the input controls are located.

The method of analysis used is stiffness method which is considered to be the most suitable method for computer implementation.

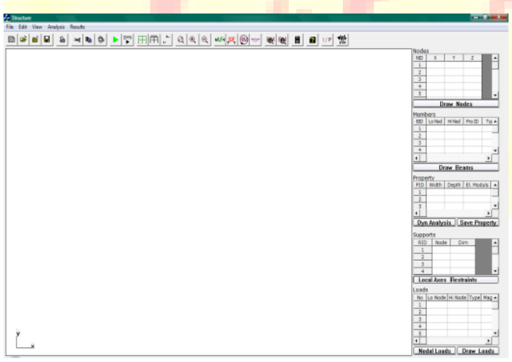

Figure 5. Layout of Analysis Module

In the window to model a frame follow these steps:

- In the 'Nodes' grid, enter the nodal values with x and y coordinate of the model.
- After entering the nodal coordinates or in between, press the button 'Draw Nodes' to draw the nodal points on the drawing area with node labels.
- In the next grid, 'Members', enter:

1. the lower Node number
2. the higher node number
3. the property id(which will be created in the next grid)
4. the type of the member (depends on end connection *viz*, 1-rigid-rigid, 2-rigid-pinned, 3-pinned–rigid, etc).

- Press the 'Draw members' button to draw the members already entered.
- In the next grid, 'Properties' enter the section dimensions and the elastic modulus and press "Save Property" to assign the property to the members corresponding to the property numbers already entered. If dynamic analysis is intended, mass in kg per unit length of the member also is to be entered in the properties grid.
- In the next grid, we can enter the support conditions.
- The next step is to enter the member loads or nodal loads in the last grid, 'Loads'. For Members enter:

1. lower node
2. higher node of the element on which the load is to be applied
3. type of the load(1- UDL, 2 – Point Load, 3 – Trapezoidal Load, etc)
4. magnitude of load
5. distance from the lower node for trapezoidal load and point load
6. press the 'Draw Loads' button.

- To enter nodal loads (point loads, point moments, if any), click on the 'Nodal Loads' button next to the 'Draw Loads' button, another grid will be displayed in the drawing area of the window and after entering the values click on 'Draw Loads' and close the popped up grid using the close button for the control.
- In between the modelling process, the model can be zoomed in or out using the middle button by scrolling.

- After successfully modelling the structure, press on the 'Analyse' button to analyse the structure.

After analysis, the bending moment, shear force, axial force and the deformed shape can be displayed using the corresponding buttons on the tool bar. Snapshot of the modelling process is shown in the figure 6.

For zooming in and out the model, use the scroll wheel of the mouse, scrolling up will zoom in the entire drawing area and scrolling down will zoom out the drawing area. There are other buttons also for facilitating the easy modelling of the structure like, show the structure only, hide/show loading, hide/show node label, etc. Another facility available is the calculator function.

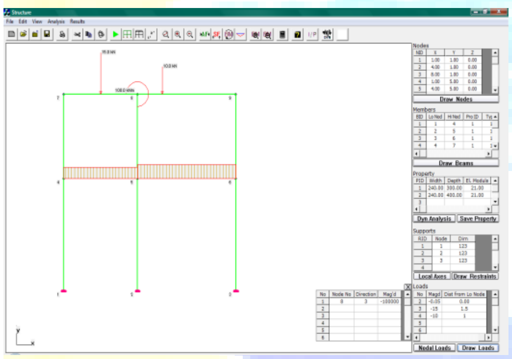

Figure 6. A frame with different loads

To display the bending moment diagram, click on the button on the toolbar. Similarly, we can display the shear and axial force diagrams.

To view the input data and the results of analysis the 'O/P' button is used and the data will be displayed in the drawing area in tabular form.

RESULTS AND DISCUSSION

The development of any programme is complete only when it gives the results exactly or within the permissible allowance as per the expectations. To verify the reliability and usefulness, the programme must be checked and compared with manual/theoretical calculations and results. If possible it can be compared with the results obtained from already available similar programmes.

To test the programme developed as part of this study, it is compared with the results from the Staad.Pro 2004, which is very popularly used designing software worldwide and with manual calculations.

Static Analysis

A frame shown below with two members is analysed using manual method, using Staad.Pro, by using the developed software and the results are compared for checking the reliability of the software developed.

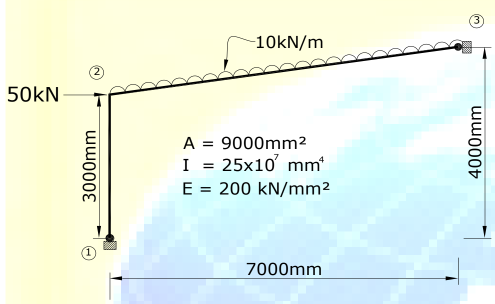

Figure 7. Frame Analysis Problem

<u>Problem:</u>

Find the element end forces for element 1-2 of the frame shown below if a uniformly distributed load of 10kN/m is applied to element 2-3 and a horizontal force of 50kN(to the right) is applied to node 2. (Figure 7).

Comparison between the values obtained from the developed software, StaadPro 2004 and manual calculations are shown in Table 1.

Dynamic Analysis

A continuous beam with two spans having the following details is shown in figure 8. Calculate the natural frequencies and mode shapes for the beam using consistent mass formulation. To reduce the problem size, axial rigidity may be assumed.

Cross sectional area, A = 0.05 m^2

Moment of Inertia , I = 1.25x 10^6 mm^4

Elastic Constant, E = 200 x 10^9 N/mm^2

Mass per unit length, m= 500 kg/m

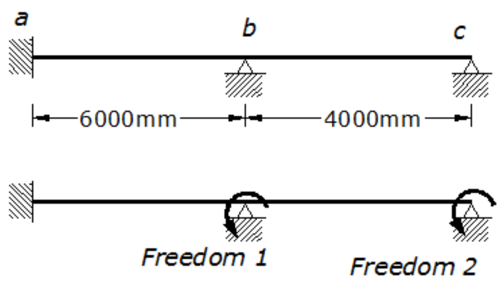

Figure 8. Problem for dynamic analysis

The results obtained are tabulated and compared with that from manual design and it is presented in Table 2. The design was almost matching with the manual calculation.

Design of Column

Design a short column with the following details and compare the results with that of StaadPro and manual calculation.

Problem Details

Member end forces:

Axial load = 2000 kN

M_{ux} = 160 kNm

M_{uy} = 120 kNm

Dimension = 400 x 600

Concrete = M20

Steel = Fe 415

Effective length =2.8m

Reinforcement is equally distributed on four sides.

The column is designed as short column with biaxial moments

20mm diameter bars with 40mm clear cover is provided.

The column was designed using StaadPro, Manual calculation and by using the developed software. The results are tabulated for comparison and presented in Table 3.

Design of Slab

An interior panel of slab with four edges continuous was designed using the manual method, StaadPro and using the developed software.

Given Data:

Grade of concrete used, f_{ck} = 20 N/mm^2

Grade of steel used, f_y = 415 N/mm^2

Clear span in shorter direction, l_x = 3.20 m

Clear span in longer direction, l_y = 4.43 m

Comparison between the values obtained from the developed software, StaadPro 2004 and manual calculations are shown in Table 4.

DESIGN OF BEAM

Moment, M_z = 160 kNm

Shear, F_y = 126 kN

The beam section is = 300x 600 mm

f_y = 415

Concrete grade = M20

Torsion is neglected.

The comparison between the values obtained from the developed software, StaadPro 2004 and manual calculations for the design of a beam are shown in Table 5.

Table 1. Comparison of results – Analysis of Plane Frame

Node	Node i			Node j		
Method	AF(kN)	SF(kN)	BM(kNm)	AF(kN)	SF(kN)	BM(kNm)
PFADS	50.890	-14.060	0.000	-50.890	14.060	-42.160
STAAD	50.804	-13.907	0.000	-50.804	13.907	-41.722
MANUAL	51.000	-14.100	-0.060	-51.000	14.100	-42.180

Table 2. Comparison of results – Dynamic Analysis of Plane Frame

	Frequency (rad/s)		Mode Shapes	
Method	$\square_1$	$\square_{\square}$	$\square_\iota$	$\square_{\square}$
PFADS	14.16	35.47	$[1.0 \quad -0.9]^T$	$[0.327 \quad 1.0]^T$
MANUAL	14.10	35.50	$[1.0 \quad -0.9]^T$	$[0.317 \quad 1.0]^T$

Table 3. Comparison of results – Column Design Module

METHOD	Longitudinal Steel Area(mm^2)	Stirrups
PFADS	3769.0 mm^2	□□ @300 mm c/c, 2 Lgd
STAAD	12, 20□.3769.9 mm^2	□□ @300 mm c/c, 2 Lgd
MANUAL	12, 20 □:3769.9 mm^2	□□ @300 mm c/c, 2 Lgd

Table 4. Comparison of results – Slab Design Module

Span	Shorter Span				Longer Span			
Location	Edge		Middle		Edge		Middle	
METHOD	Area (mm^2)	Spacing (mm)	Area (mm^2)	Spacing (mm)	Area (mm^2)	Spacing (mm)	Area (mm^2)	Spacing (mm)
PFADS	179.0	8,260	144.0	8,260	144.0	8,260	144.0	8,260
STAAD	172.0	8,290	144.0	8,300	144.0	8,270	144.0	8,270
Manual	179.5	8,280	167.5	8,300	167.5	8,300	167.5	8,300

Table 5. Comparison of results – Beam Design Module

METHOD	Tension Steel Area(mm^2)	Stirrups
PFADS	906.8 mm^2	□□ @300 mm c/c, 2 Lgd
MANUAL	900.0 mm^2	□□ @300 mm c/c, 2 Lgd

The developed software was checked for standard problems of both design and analysis and the results obtained are tabulated accordingly for comparison with manual calculation and with another programme, StaadPro 2004. Looking into the results from the developed software and the results from StaadPro 2004, the results are almost identical. Also the manual calculations show that the values obtained from the developed software are closer to the exact values. This shows the accuracy of the developed software and close matching with the theoretical manual calculations ascertains the reliability of the software developed.

Advantages of the Software

The developed software has the following advantages:

1. The software provides exact requirements without the complexities and confusion of so many windows.

2. Unlike other design packages, the design modules contain only one window in which only the necessary controls are arranged so that even a new user will not feel any difficulty in using the software.

3. It is very particular in the sense of its application – The analysis module is for the structural analysis of plane frames only, without much complications.

4. Once the additional facilities like, member selection, 3D modelling and analysis facilities, etc, are added to the software, it can replace the commercially available packages which are very costly also.

5. The column design module has built in interaction diagram for visually assessing the safety of the column.

Limitations of the Software

The limitations of the software are:

1. The analysis module at present can analyse only plane frames but without too much modifications, it can be turned into a full fledged 3D analysis programme including dynamic analysis.

2. The design package relies upon the accuracy of the loading pattern which is to be collected from any other analysis package.

3. Design package at present can handle only the three components, viz, columns with biaxial bending, rectangular beams and rectangular slabs.

CONCLUSIONS

During the last few decades, computer software has become more and more critical in the analysis of engineering and scientific problems. Much of the reason for this change from manual methods has been the advancement of computer techniques developed by the research community and, in particular, universities.

The following are the drawn up conclusions that have emanated from the research and implementation of this project:

A user-friendly program for the computer analysis of building frames and reinforced concrete design of beams, slabs and columns has been developed and tested, and the study emphasises the necessity of knowledge in the theory behind structural analysis and design programs to enable civil engineers to

- Idealise structures in a sensible manner
- Prepare the input data, knowing at least, in general terms, how it will be used by the computer.
- Interpret the output generated by the computer
- Know when and why a structure is beyond the capabilities of a computer program and
- Devise cross checks when problems arise.

REFERENCES

[1] Sujith Kumar Roy and Subrata Chakrabarty, *Fundamentals of Structural analysis*, S.Chand Publications, New Delhi, 2003.

[2] IS: 456 – 2000 *Indian Standard Plain and Reinforced Concrete – Code of Practice.*

[3] SP 16 – 1980, Design *Aids for Reinforced Concrete to IS: 456 – 1978*