SLAB ANALYSIS AND DESIGN SOFTWARE BASED ON ACI CODE

Qaisar Ali*, Qaiser Hayat*

ABSTRACT

This paper describes, ESSAD (Edge Supported Slab Analysis and Design) Ver 1.0 software based on the ACI code. Commercially available packages for slab analysis and design based on finite element methods are generally demanding for structural engineers as they require extensive knowledge and background in Structural Analysis. However, ESSAD follows the ACI code which is much easier to understand and is widely used as an approximate but sufficiently reliable method for slab analysis and design problems. Therefore, while using ESSAD, an engineer knows precisely what the software is going to perform with the data it requires and what are its limitations and approximations for a specific case. This is extremely important as without knowing the mechanism and limitations of a slab analysis and design package, its use can pose problems.

INTRODUCTION

ESSAD 1.0 is a simple and user-friendly package for slab analysis and design based entirely on the ACI code. Creation and modification of a model, execution of the analysis, and checking and optimization of the design are all done through a single Graphical User Interface (GUI). Tabulated displays and hard copies of the results can be easily produced.

ESSAD 1.0 is structured to support the ACI code throughout its operation, whether it is analysis or design. Manual use of the ACI code is difficult and time consuming for a number or reasons. Firstly, a vast number of tables ad graphs are consulted in this method and an enormous number of checks are to be applied each time. Extreme care is to be taken while selecting values of slab moment and shear coefficients, calculating steel areas, etc. which if done incorrectly may affect the ultimate results. Besides this, every now and then interpolations have to be carried out. Secondly, although the analysis is approximate, it takes a large amount of time to solve the slab systems of a several story building and a tremendous amount of data comes into play which can result in errors. ESSAD 1.0 can be used effectively to address these problems. Once data, i.e. a slab system’s dimensions, properties, and loading is fed into the package, the package automatically takes care of the end conditions and the case number. It selects the correct values from the tables ad interpolates them if necessary. Moments and bar areas are determined with all necessary checks applied.

DEVELOPMENT PLATFORM

Visual Basic is a Rapid Application Development (RAD) platform that is not only simple to use but also makes excellent graphical user interfaces providing a simple and effective dialogue with the designer.

USING THE SOFTWARE

ESSAD 1.0 analyzes and designs a slab system using a model that the user defines with the graphical user interface. The model may include the following features which represent the basic structure:

- Material properties
- Nodes; which act as a base to assign slab panels
- Loads; including self-weight, dead weight, live weight

After ESSAD 1.0 analyzes the slab system, it records the slab connectivity i.e. which two slabs are interconnected and stores that data in collections (a special type of array).

* Department of Civil Engineering, NWFP University of Engineering and Technology, Peshawar.
The graphical user interface provides the designer with many powerful features to create the model. Starting with a single panel or multi-panel slab, the user can define the entire model by simply entering the slab dimension and number of panels in the two axes. In case of a complicated slab system i.e. one with dissimilar slab panels, the user can use a slab grid control. This is a very powerful feature, and overly simplifies the creation of model for a complicated slab system.

ASSIGNING SLAB PANELS

Slab panels are assigned specific positions (four nodes) to draw the model according the actual dimensions.

Assigning is done either:

a. Through the mouse by clicking four successive nodes on which the slab panel will rest, in a Z pattern as shown in Fig. 1. Actually the slab panel rests on beams rather than on nodes, but ESSAD 1.0 recognizes nodes as the four objects on which the slab panel rests on its four corners.

b. Through the keyboard by entering four successive node numbers in the text boxes.

DATA INPUT

The materials are input through GUIs as shown in Fig. 2. The loads are specified through the Loading menu. If similar loads are to be applied on all panels, a single click can automatically assign similar loads to all panels.

ANALYSIS

After the user has created a complete structural model, the model can be analyzed to determine the resulting moments by choosing this option from the Main Menu Bar or by choosing the short cut button from the Main Tool Bar.

The program performs the following tasks during the analysis phase:

a. Determines the end conditions.
b. Determines the Case Number (according to the ACI Code).
c. Determines the moment coefficients by using the available three tables.
d. Determines the moments at five different points in both directions for each slab panel by using the respective moment coefficients, dimensions and loading on the slab panel.
e. Save the analysis data in different collection.

DESIGN

After the successful execution of analysis phase, the design process can be activated by choosing the design option from the Main Menu Bar.

The program performs the following tasks during the design operation:

a. Determines the minimum slab depth according to the ACI code.
b. Determines the values of minimum and maximum steel areas.
c. Determines the design moment for minimum area of steel. If it is less than the ultimate moment, a design moment greater than the ultimate moment is chosen and area of steel is determined for that moment.
d. Determine the spacing and bar number for the area of steel determined earlier.

e. Checks are applied on the steel bar spacing and the area of steel.

DISPLAYING RESULTS

The Analysis Output is displayed in Tabular form as shown in Fig. 3.

Fig. 4 shows a sample design output showing steel bars and spacing. Whereas, the detailed design outputs are viewed as a Tree view display (grouping similar things under common headings) are shown in Fig. 3.

ALGORITHM

The Model is completely defined by specifying the dimensions, material properties, loading and end conditions of each slab panel.

The whole Slab System is analyzed by analyzing each slab panel one by one, using the ACI Coefficients for dead load and live load, for both the negative and positive moments, and also using the loads for each slab panel. The coefficients are chosen from a database in which the ACI Coefficient tables are stored.

Design is done by using the positive and negative moments obtained as consequence of a successful analysis.

The results are stored in collections and are used to produce tabulated and graphical output according to the requirement of the user.

CONCLUSIONS

ESSAD 1.0 has remarkably reduced the work involved in solving edge supported slab systems. This makes it possible to perform the slab analysis and design for extremely length problems such as multi-storey buildings with a minimum effort.
Fig. 3. Sample Screen Shot of Analysis Output

Fig. 4. Sample Screen Shot of Graphical Design Output Steel Bars and Spacing.
Fig. 5. Sample Screen Shot of Design Details

REFERENCES


Reinforced Concrete; Mechanics and Design.
