A COMPUTER SOFTWARE

A computer program for analyzing two- and three-dimensional framed structures is available on the publisher's website www.cengage.com/engineering. The software, which can be used to analyze plane and space trusses, beams, plane and space frames, and grids, is based on the matrix stiffness method. It can also perform geometrically nonlinear analysis of plane trusses. The software is designed for use on IBM and IBM-compatible personal computers with Microsoft Windows® operating systems, and it provides an option for saving input data into files for subsequent modification and/or execution.

Complete instructions for downloading and installing the software are provided on the publisher's website www.cengage.com/engineering/kassimali.

Starting the Computer Software

1. Click the **Start** button on the taskbar.
2. Point to the menu title **Programs** and then click the menu item **MATRIX ANALYSIS OF STRUCTURES 2.0—Kassimali**; the software's title screen will appear.

Inputting Data

The computer software is designed so that any *consistent* set of units may be used. Thus, all the data must be converted into a consistent set of units before being input into the software. For example, if we wish to use units of kips and inches, then the joint coordinates must be defined in inches, the moduli of elasticity in ksi, the cross-sectional areas in $in.^2$, the moments of inertia in $in.^4$, the joint loads and moments in kips and k-in., respectively, and distributed member loads in k/in.

To start entering data for a structure, click the menu title **Project**; and then click the menu item **New Project**. The input data necessary for the analysis of a structure is divided into six categories; the data in each category is input by clicking on the corresponding menu title and then entering information in the forms and/or dialog boxes that appear on the screen. The input data categories are:

1. General structural data (project title and structure type)
2. Joint coordinates and supports
3. Material properties
4. Cross-sectional properties

5. Member data (beginning and end joint numbers, material and cross-sectional property numbers, member hinges if applicable, and angle of roll in the case of space frames)
6. Loads (joint and member loads, support displacements, temperature changes, and fabrication errors)

Results of the Analysis

Once all the necessary data has been entered, click the menu title **Analyze** of the main screen to analyze the structure (Fig. A.1). The software will automatically compute the joint displacements, member end forces, and support reactions, using the matrix stiffness method. The results of the analysis are displayed on the screen. The input data as well as the results of the analysis can be printed by clicking on the menu title **Project** and then clicking on the menu item **Print**, of the main screen.

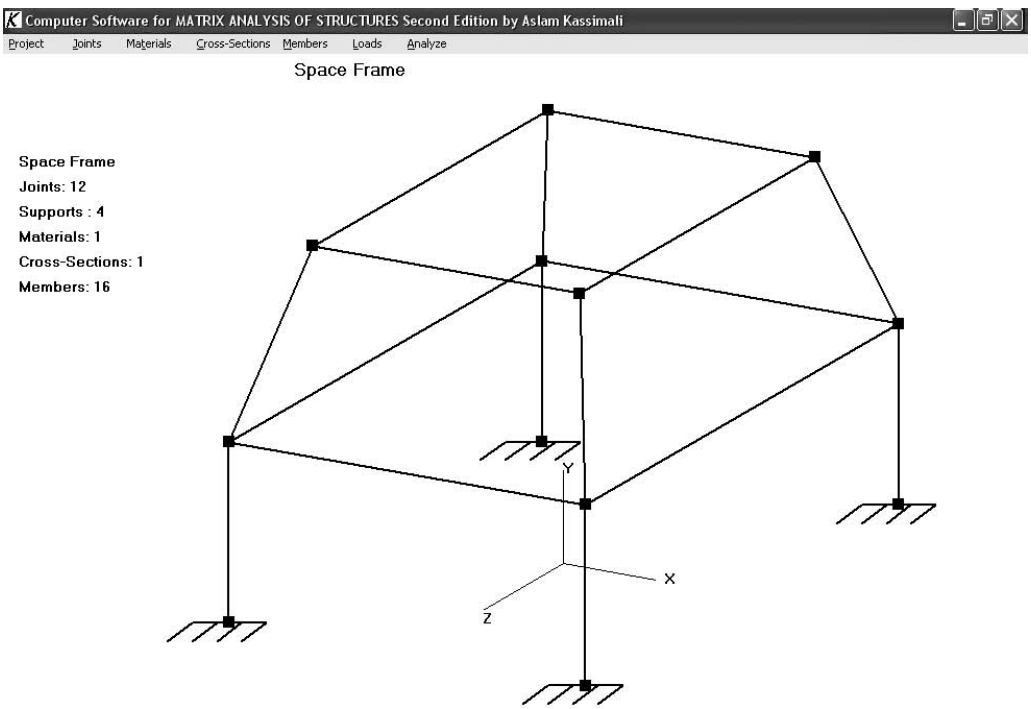

Fig. A.1 Main Screen