
SOFTWARE DEVELOPMENT :

F.E.A.P (FINITE ELEMENT ANALYSIS PROGRAM)

A project report presented in partial fulfillment
of the requirement for the award of
Advanced Diploma In Civil Engineering of
MARA Institute of Technology.

by:

AHMAD KAMAL BIN ABDUL AZIZ

**DEPARTMENT OF CIVIL ENGINEERING
MARA INSTITUTE OF TECHNOLOGY
SHAH ALAM 40450, SELANGOR**

MEI 1992

Synopsis

Finite element techniques is known exclusively to solve structural engineering problems. Most of the existing computer software has been in the form of large software packages and very few for general use. This project is an attempt to make the techniques available to a much larger population at very minimal cost.

ACKNOWLEDGEMENT

The writer wish to express his gratitude to Puan Afidah Abu Bakar under whose supervision the work of this project was undertaken and whose continued aid, guidance, advice and encouragement has been invaluable throughout the project.

The writer would also like to thank all personnels especially his wife who have given their time and support .

TABLE OF CONTENTS

	Page
SYNOPSIS	i
ACKNOWLEDGEMENT	ii
TABLE OF CONTENTS	iii
LIST OF FIGURES	v
NOTATION	vi
<u>CHAPTERS</u>	
1.0 INTRODUCTION	
1.1 OBJECTIVE AND SCOPE OF WORK	1
1.2 REVIEW OF PAST WORK	2
2.0 INTRODUCTION TO FINITE ELEMENT METHOD	
2.1 GENERAL IDEA	3
2.2 IMPORTANT CONCEPTS	4
2.3 GENERAL FINITE ELEMENT ANALYSIS PROCEDURE	6
2.4 BENEFITS FROM USING THE FEM	7
3.0 FINITE ELEMENT ANALYSIS FOR 2-D PLANE STRESS/STRAIN	
3.1 INTRODUCTION	8
3.2 DEVELOPMENT OF EQUATIONS	10
3.3 IMPLEMENTATION OF PROGRAM	20
3.4 PROGRAM VERIFICATION	24
4.0 FINITE ELEMENT ANALYSIS FOR PIN-JOINTED PLANE FRAME.	
4.1 INTRODUCTION	33

1.0 INTRODUCTION

1.1 OBJECTIVE AND SCOPE OF WORK

The main objective of the project is to develop a finite element program that are interactive, menu driven and user friendly. The developed program called "FEAP" (i.e Finite Element Analysis Program) are capable of analysing isotropic plate subjected to loads, pin-jointed plane frame and two dimensional plane stress and plane strain structural problems.

FEAP program consist of several programs which are linked together to produce a comprehensive software for the analysis of structure. The program allows the user to input structure configuration and loading data in computer aided mode. The response output can be obtained on the screen or via the printer. The output results are well presented and well organised.

The program is developed for the IBM and compatible microcomputer system. Programs were developed using Microsoft Quickbasic (Version 4.00) Compiler. The development of the program is in response to the need for greater utilization of microcomputer application since most finite element software packages are only applicable