

NONLINEAR ANALYSIS OF REINFORCED

CONCRETE BEAM : A Case Study

by:

ZULKORNAIN BIN YUSOFF

**A Report Submitted To School Of Civil Engineering, Mara Institute Of
Technology, Shah Alam In Partial Fulfilment Of The Requirement For
The Award Bachelor Of Engineering (Hons.) (Civil)**

NOVEMBER 1996

ACKNOWLEDGMENTS

I wish to convey my sincere thanks to my supervisor, Pn Afidah binti Hj. Abu Bakar for the supervision, guidance, advices and encouragement and criticism throughout the course of the work. I would also like to thanks Dr. Azmi Ibrahim for the discussions which have been conducted and the practical information received throughout the project regarding ANSYS programming operations.

I would like to thanks Computer Aided Design Engineering and Manufacturing (CADEM) Staff for their cooperation in using CADEM facilities with regard in this study.

Last but not least, I would like to thanks my family and all my friends who had given me their moral support to complete this project.

ZULKORNAIN BIN YUSOFF

November 1996

TABLE OF CONTENTS		PAGE
ACKNOWLEDGMENTS		i
TABLE OF CONTENTS		ii
LIST OF FIGURES		iv
LIST OF TABLES		v
LIST OF PLATES		vi
LIST OF SYMBOLS		vii
ABSTRACT		viii
CHAPTER 1 : INTRODUCTION		
1.1	General	1
1.2	Scope Of Work	2
1.3	Objectives	3
1.4	Outline Of The Report	3
CHAPTER 2 : NONLINEAR FINITE ELEMENT ANALYSIS		
2.1	Background	4
2.2	Nonlinear Material Behaviour	5
2.3	Basic Nonlinear Solution Technique	5
2.3.1	Iterative Procedures	7
2.4	Finite Element (FE) Modelling	10
2.4.1	The Concrete Model	10
2.4.2	Material Behaviour	11
2.4.3	Uniaxial Condition	12
2.4.4	Reinforce Steel Model	13
2.5	Tensile Cracks	14
2.6	Numerical Modelling Of Cracks	14
2.6.1	Smearred Crack Method	15
CHAPTER 3 : FINITE ELEMENT ANALYSIS USING ANSYS		
3.1	How Program Is Organise	16
3.1.1	Begin Level	17
3.1.2	The Processor Level	17
3.2	Idealisation Model	17
3.2.1	Build The Model	19
3.2.2	Model Material Nonlinearities	20
3.2.3	Definition Of Model	20
3.2.4	Boundary Condition	21
3.2.5	Application Force	22
3.2.6	Load Step Option	22
3.3	Review The Results	24

ABSTRACT

A number of material-related factors can cause stiffness of structure to change during the course of analysis. Nonlinear stress-strain relationship of plastic will cause a structure stiffness to change at different load levels.

In this study, the nonlinear finite element analysis of a reinforced concrete beam using ANSYS (Swanson Analysis System), a finite element analysis system is presented. ANSYS element was utilised to model the beam in three dimension (3D) were adopted in the analysis. Two case of support conditions were adopted in the analysis, i.e. firstly considering no anchorages and secondly with proper anchorage at the end joints. Crack propagation was also studied on the beam.

1.0 INTRODUCTION

1.1 General

The search for a way to represent the true nonlinearity of structure goes back to renaissance times, and present theories of nonlinear elastic and inelastic behaviour are the result of approximately two hundred years of steady development. But only recently has computer made it possible to put much knowledge to use in design.

The finite element method, simulated by rapid growing power of electronic digital computers, has strengthened its position in computer in computational mechanic from its early stage of performing linear analysis for simple problem into an era of conducting the more challenging tasks, such as nonlinear, inelastic dynamic analyses and so on, for problem that are ten, hundred, or thousand times more complicated than those ever attempted before. Nowadays, many people tend to view the finite element method as a well-established tool that can be readily applied to the solution of various nonlinear problems. Experience has shown us that can be too optimistic in many cases , as the various phases involved in a nonlinear analysis are not as definite and straightforward as those of linear analysis.(Yang, Yeong -Bin, 1954).