

HIGHER TECHNICAL INSTITUTE

MECHANICAL ENGINEERING DEPARTMENT

DIPLOMA PROJECT

**IMPROVEMENTS OF VIBRATING STRUCTURE
USING FINITE ELEMENT ANALYSIS**

By

NEOPTOLEMOU MICHAEL (M/766)

JUNE 1996

**HIGHER TECHNICAL INSTITUTE
MECHANICAL ENGINEERING COURSE**

DIPLOMA PROJECT

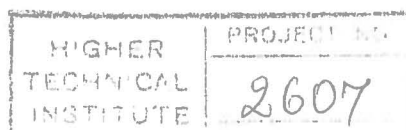
**IMPROVEMENTS OF VIBRATING STRUCTURES USING
FINITE ELEMENT ANALYSIS**

Student: Neoptolemos Michael (3ME2)

Supervisor: Dr. A. Stassis

External Examiner: G. Kampouris

Project Number: M/766



**IMPROVEMENTS OF VIBRATING STRUCTURES
USING FINITE ELEMENT ANALYSIS**

by

Michael Neoptolemou

Project Report

Submitted to the Department of Mechanical

Engineering of

HTI

Nicosia, Cyprus

In partial fulfillment of the requirement

for the Diploma of

TECHNICIAN ENGINEER

in

MECHANICAL ENGINEERING

June 1996

HIGHER TECHNICAL INSTITUTE	PROJECT NO. 2607
----------------------------------	---------------------

ACKNOWLEDGMENTS

My supervisor on this Diploma Project, has been Dr. A. C. Stasis, Lecturer in the Mechanical Engineering Department of the Higher Technical Institute, to whom I own a great debt of gratitude for the guidance, support and encouragement he has afforded me throughout the completion of this project.

I also acknowledge the valuable guidance and assistance received from my father Mr. A. Neoptolemou in the sourcing and securing of mechanical parts and spares required for the tests, experiments and simulations undertaken throughout this project.

TABLE OF CONTENTS

	page
ACKNOWLEDGMENTS	iv
CONTENTS	v
SUMMARY	xiii
INTRODUCTION	xiv
CHAPTER 1 - THE FINITE ELEMENT METHOD	1
1.1 Introduction	1
1.2 Advantages and disadvantages of FEM	1
1.3 Aspects of FEM	2
1.3.1 Discretization of the domain	3
1.3.1.a One-dimensional elements	3
1.3.1.b Two-dimensional elements	4
1.3.1.c Three-dimensional elements	5
1.3.2 The division of the domain into elements	6
1.4 Linear interpolation Polynomials	7
1.5 Interpolating Polynomials for a Discretized region	7
1.6 Finite Element formulation	7
1.7 Torsion of Noncircular sections	8
1.8 Computer Implementation	8

1.9	Heat transfer by conduction and convection.....	10
1.10	Fluid Mechanics: International flow.....	10
1.11	Radial and Axisymmetric field problems.....	10
1.12	Time Dependent field problems.....	11
1.13	Solid Mechanics: Elasticity.....	11
1.14	Higher order Elements: Computer Implementation.....	11

CHAPTER II - BASICS ON FINITE ELEMENT ANALYSIS.....12

2.1	Introduction.....	12
2.2	Differences between classical methods and finite elements.....	12
2.3	Steps of finite element analysis.....	13

2.3.1	Step 1: Discrete or model the structure.....	13
2.3.2	Step 2: Define the element properties.....	14
2.3.3	Step 3: Assemble the element stiffness matrices.....	14
2.3.4	Step 4: Apply on the loads.....	14
2.3.5	Step 5: Define boundary conditions.....	14
2.3.6	Step 6: Solve the system of linear algebraic equations.....	14
2.3.7	Step 7: Calculate stresses.....	15

2.4 Example illustrating all the steps of a typical FEA.....15

2.4.1	Objectives of the example.....	15
2.4.2	Static analysis.....	16
2.4.3	Dynamic analysis.....	21

CHAPTER III - COMPUTER-AIDED DESIGN27

3.1 Introduction.....27

3.2 Cad capabilities.....27

3.3 Applications in Mechanical Design.....29

3.4 Modeling.....29

3.4.1 Wireframe models.....34

 3.4.1.1 Limitations of Wireframe modeling.....34

3.4.2 Surface models.....35

 3.4.2.1 Advantages of surface modeling over
 the wireframe.....35

 3.4.2.2 Types of surfaces modeling.....36

3.4.3 Solid models.....36

 3.4.3.1 The (CSG) constructive solid geometry
 the (b-rep) boundary representation.....36

 3.4.3.2 Advantages of solid modeling.....37

CHAPTER IV - FEA ON A PC.....38

4.1 The process of doing FEA on a PC.....38

4.2 Algor for Mechanical engineering design.....38

4.3 Structural considerations.....40

4.4 Modeling and planning.....40

4.5 Creating new designs.....40

4.6 Redesigning existing parts.....41

CHAPTER V - IMPROVEMENTS OF A VIBRATING CAR

CHASSIS USING FEA.....42

5.1 Introduction.....42

5.2 Conventional chassis.....43

5.3 Chassis frame.....50

5.4 The analysis.....60

CHAPTER VII - CONCLUSION AND DISCUSSION.....99

REFERENCES.....102

APPENDICES.....104

TABLE OF FIGURES

		page
FIGURES		
1.3.1	Some one-dimensional finite elements.....	4
1.3.2	Some two-dimensional finite elements.....	5
1.3.3	Some three-dimensional finite elements.....	6
2.2.1	Cylindrical tank modeled with plate elements.....	13
2(a)	Axially loaded bar.....	16
2(b)	Finite element model.....	16
3	Two-degree-of freedom truss element.....	16
4	Axial deformation of truss element.....	17
5	Element modal elastic forces and displacements.....	19
6	Free-body diagram of elements and nodes.....	19
7	Lumped mass discretization of truss element.....	22
8	Elastic and Inertia element nodal forces.....	23
9	Element nodal dynamic forces and displacements.....	23
3.4.1	Wireframe model.....	30
3.4.2 (a)	Finite-element analysis: FEM of part shown.....	32
3.4.3 (b)	FEM showing stress contours.....	33
5.2.1	Chassis-cum body construction.....	44
5.2.2	AUSTIN 8 CHASSIS.....	44
5.2.3	INSWEPT CHASSIS.....	45
5.2.4	Heavy commercial vehicle chassis.....	46
5.2.5	BMW solid modeling of a "3" series car.....	47
5.2.6	MAZDA's development policy.....	48
5.2.7	BMW FEA of a "3" series car.....	49

5.3.1	A front frame by Mercedes-Benz.....	52
5.3.2	A front frame of a Mazda 626.....	52
5.3.3	Golf's test about chassis stiffness.....	53
5.3.4	Some application of FEA in automotive industry.....	55
5.3.5	Some applications of FEA.....	56
5.3.6	Crushable body construction by Mazda.....	57
5.3.7	Computer-simulated bending rigidity of the bodyshell by Mazda.....	58
5.3.8	Computer-simulated rigidity of the bodyshell by Mazda....	59
6.1	FEA mesh with 2871 tetrahedral elements without boundary conditions	61
6.2	FEA mesh with boundary conditions.....	61
6.3	Plan view of the chassis including all necessary dimensions.....	62
6.4	Side view of the chassis.....	63
6.5	Back view of the chassis.....	64
6.6	Plan view of the FEA mesh with tetrahedral plate elements with 3mm mild steel thickness.....	67
6.7	Back view of the FEA mesh.....	67
6.8	FEA mesh with tetrahedral plate elements with 3mm thickness mild steel	68
6.9	Dynamic modal analysis of the chassis at the first natural requency.....	72
6.10	Second frequency Dynamic modal analysis.....	73
6.11	Third frequency Dynamic modal analysis.....	74
6.12	Fourth frequency Dynamic modal analysis.....	75

6.13	Fifth frequency Dynamic modal analysis.....	76
6.15	Sixth frequency Dynamic modal analysis.....	77
6.16	First frequency Dynamic modal analysis of the first modified chassis.....	80
6.17	Second frequency Dynamic modal analysis of the first modified chassis.....	80
6.18	Third frequency Dynamic modal analysis of the first modified chassis.....	81
6.19	FEA mesh with boundary conditions.....	82
6.20	Plan view of the third modified chassis including all necessary dimensions.....	84
6.21	Back view of the modified chassis.....	85
6.22	Side view of the modified chassis.....	86
6.23	First frequency Dynamic modal analysis of the second modified chassis.....	89
6.24	Second frequency Dynamic modal analysis of the second modified chassis.....	90
6.25	Third frequency Dynamic modal analysis of the second modified chassis.....	91
6.26	First frequency Dynamic modal analysis of the third modified chassis.....	95
6.27	Second frequency Dynamic modal analysis of the third modified chassis.....	96
6.28	Third frequency Dynamic modal analysis of the third modified chassis.....	97

6.29	Third frequency Dynamic modal analysis of the third modified chassis with a side view.....	98
------	--	----

TABLES

6.1	Material characteristics of the chassis.....	66
6.2	Algor dynamic modal analysis	71
6.3	Algor dynamic modal analysis of the first modified chassis.....	79
6.4	Algor dynamic modal analysis of the second modified chassis.....	88
6.5	Algor dynamic modal analysis of the third modified chassis.....	94

SUMMARY

The finite element method is a computational technique based on numerical procedure which produces 3D modeling of structures allowing engineers to analyze stress factors within the structure when a variety of forces, vibrations and conditions are applied to the structure as a whole or any particular element of this structure.

Several experiments were performed on a software called Algor to test a car chassis for the stresses applied on it during the engine running. Simulations and photos have been taken and an improvement procedure was followed to develop the mathematical model under structural modifications.

The results obtained with a finite element analysis, in my opinion, can be very accurate when a proper finite element model is used. The engineer who wishes to develop a finite element code, must understand the finite element theory, first. The engineer, however, who wants only to use a finite element code, can work by simply having: rudimentary understanding of the fundamental concepts of the finite element method and training of the computer program that will be used.

INTRODUCTION

The finite element analysis, is a powerful procedure for solving the mathematical problems of engineering and physics. Its applications ranges from the analysis of a structural framework of an aircraft or an automobile to that of a fluid flowing through a duct, over a weir, or through the earth.

Actually, the FEA is a numerical procedure for solving the differential equations of physics and engineers. In structural problems, the method produces a set of linear equilibrium equations by minimizing of the FEA with a minimization procedure quickly led to its eye in other engineering areas. The method was applied to problems governed by Laplace or the Poisson equations because are closely related to the minimization of a functional.

What makes a finite element model finite is that you take the part you want to analyze and break it down into a finite number of chunks, called elements. Elements are connected to each other at their boundaries by points called nodes and contain a mathematical description of the material properties of the part. The nodes and elements connected together form a mesh, which looks like the part being modeled. There are two types of elements called h and p. An h-element mesh is refined by adding more h-elements, while a p-element mesh is refined by increasing the order of the displacement polynomial in the p-elements.

The applications of this method began with the conduction heat transfer analysis, then the fluid mechanics followed and finally the method was applicable to any system which was including differential equations.

The finite element method was developed because of the need for a more accurate analysis of aircraft frames and the national commitment to space exploration. The digital computer provided a rapid means of performing the many calculations involved.

As is been reported the FEA can be applied in a large variety of other things. The mechanical engineering is one of the many things. In our days, where the hi-technology is dominating the world, every single product which is produced is tested. As the technology is developed the demands of the customers are increased. No excuses can take place in our days. The people are very predicative in breakdown situations. So, every company all over the world has software. In our case with Algor's capabilities we can test even a coffee-machine.