

*Using ANSYS for Finite Element Analysis: Dynamic, Probabilistic
Design and Heat Transfer Analysis, Volume II*

Copyright © Momentum Press®, LLC, 2018.

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means—electronic, mechanical, photocopy, recording, or any other—except for brief quotations, not to exceed 400 words, without the prior permission of the publisher.

First published by Momentum Press®, LLC
222 East 46th Street, New York, NY 10017
www.momentumpress.net

ISBN-13: 978-1-94708-322-6 (print)
ISBN-13: 978-1-94708-323-3 (e-book)

Momentum Press Sustainable Structural Systems Collection

Collection ISSN: 2376-5119 (print)
Collection ISSN: 2376-5127 (electronic)

Cover and interior design by Exeter Premedia Services Private Ltd.,
Chennai, India

10 9 8 7 6 5 4 3 2 1

Printed in the United States of America

ABSTRACT

Finite Element Method (FEM) is a well-established technique for analyzing the behavior and the response of structures or mechanical components under static, dynamic, or thermal loads. Over the past two decades the use of finite element analysis as a design tool has grown rapidly. Easy to use commercial software have become common tools in the hands of students as well as practicing engineers. The objective of this two volume book is to demonstrate the use of one of the most commonly used Finite Element Analysis software, ANSYS, for linear static, dynamic, and thermal analysis through a series of tutorials and examples. Some of the topics and concepts covered in these tutorials include development of beam, frames, and grid equations; 2-D elasticity problems; dynamic analysis; and heat transfer problems. We are hoping these simple, yet, fundamental tutorials will assist the users with the better understanding of finite element modeling, how to control modeling errors, the safe use of the FEM in support of designing complex load bearing components and structures. There are many good textbooks currently used for teaching the fundamentals of finite element methods. There are also detailed users manuals available for commercial software (ANSYS). However, those sources are useful for advanced students and users. Therefore, there was a need to develop a tutorial that would supplement a course in basic finite element or can be used by practicing engineers who may not have the advanced training in finite element analysis. That is the gap addressed by this book.

KEYWORDS

ANSYS, composite materials, Dynamics, Failure analysis, Fatigue loads, FEM, optimization, statistics

CONTENTS

LIST OF FIGURES	ix
PREFACE	xi
1 DYNAMIC ANALYSIS	1
1.1 Tutorial 1: Harmonic Analysis of Structure	1
1.2 Tutorial 2: Modal Analysis of Structure	11
2 COMPOSITE MATERIALS	19
2.1 Composites—A Basic Introduction	19
2.2 Modeling Composites Using ANSYS	26
2.3 Tutorial 3: Simply Supported Laminated Plate Under Pressure	38
3 PROBABILISTIC DESIGN ANALYSIS	69
3.1 Probabilistic Design	69
3.2 Probability Distributions	75
3.3 Choosing a Distribution for a Random Variable	91
3.4 Probabilistic Design Techniques	96
3.5 Postprocessing Probabilistic Analysis Results	98
3.6 Tutorial 4: Probabilistic Design Analysis of Circular Plate Bending	107
4 APDL PROGRAMMING	145
4.1 Create the Analysis File	145
4.2 Tutorial 5: Stress Analysis of Bicycle Wrench	148
4.3 Tutorial 6: Heat Loss from a Cylindrical Cooling Fin	155

5	DESIGN OPTIMIZATION	167
5.1	Optimum Design	167
5.2	Design Optimization Using ANSYS	172
5.3	Tutorial 7: Design Optimization Tutorial	193
	BIBLIOGRAPHY	217
	ABOUT THE AUTHORS	219
	INDEX	221

LIST OF FIGURES

Figure 2.1.	Illustrating the combined effect on modulus of the addition of fibers to a resin matrix.	21
Figure 2.2.	Illustrates the tensile load applied to a composite body.	22
Figure 2.3.	Illustrates the compression load applied to a composite body.	22
Figure 2.4.	Illustrates the shear load applied to a composite body.	23
Figure 2.5.	Illustrates the loading due to flexure on a composite body.	23
Figure 2.6.	Tensile strength of common structural materials.	24
Figure 2.7.	Tensile modulus of common structural materials.	25
Figure 2.8.	Specific tensile strength of common structural materials.	25
Figure 2.9.	Specific tensile modulus of common structural materials.	26
Figure 2.10.	Layered model showing dropped layer.	29
Figure 2.11.	Sandwich construction.	32
Figure 2.12.	Layered shell with nodes at midplane.	32
Figure 2.13.	Layered shell with nodes at bottom surface.	33
Figure 2.14.	Example of an element display.	36
Figure 2.15.	Sample LAYPLOT display for [45/-45/-45/45] sequence.	37
Figure 3.1.	The flow of information during a probabilistic design analysis.	75
Figure 3.2.	The normal probability density function (pdf).	77

Figure 3.3.	The normal cumulative distribution function (cdf).	78
Figure 3.4.	The uniform probability density function.	83
Figure 3.5.	The uniform cumulative distribution function.	84
Figure 3.6.	The lognormal probability density function for four values of σ .	86
Figure 3.7.	The lognormal cumulative distribution function for four values of σ .	87
Figure 3.8.	The Weibull probability density function for four values of γ .	89
Figure 3.9.	The Weibull cumulative distribution function for four values of γ .	90
Figure 3.10.	The graph of X_1 and X_2 illustrating bad sample distribution.	97
Figure 3.11.	The graph of X_1 and X_2 illustrating good sample distribution.	98
Figure 3.12.	The cumulative distribution function of the random property X .	100
Figure 5.1.	Optimization tree listing the optimization methods.	171
Figure 5.2.	Optimization data flow.	175

PREFACE

Finite element method (FEM) is a well-established technique for analyzing the behavior of mechanical and structural components of systems. In recent years, the use of finite element analysis (FEA) as a design tool has grown rapidly. Easy-to-use commercial software have become common tools in the hands of students as well as practicing engineers.

In the first volume of this tutorial, we demonstrated the use of ANSYS for Static Analysis of solid structures. In this volume we introduce the following applications:

- Dynamic Analysis
- Composite Materials
- Probabilistic Design
- Heat Transfer
- Design Optimization Problems.

The main objective of this book is to serve as a practical tutorial to help the readers gain insight into appropriate use of finite element modeling, understand how to control modeling errors, benefit from hands-on exercise at the computer workstation, and understand the safe use of the FEM in support of designing complex load-bearing components and structures. There are many good textbooks already in existence that cover the theory of FEMs. Similarly, there are detailed user's manuals available for commercial software (ANSYS). But, those are useful for advanced students and users. Therefore, there was a need to develop a computer session manual in line with the flow of a course and utilizing a software platform, ANSYS, that is available in most engineering schools. Students will be able to acquire the required level of understanding and skill in modeling, analysis, validation, and report generation for various design problems.

This book could also be very helpful for the students of senior design (Mechanical System Design) and FEA for Large Deformation Problems.

In addition, it could be used for computer sessions of short courses on stress analysis techniques and FEA offered by Mechanical Engineering departments.

After giving a brief introduction to the finite element analysis and modeling, various guided examples have been included in this book. Several new tutorials have been developed and others adapted from different sources including ANSYS manuals in 2-volumes, ANSYS workshops, and *Internet* resources. Tutorials have been arranged in each volume according to the flow of the course and covered topics, such as solid modeling using 2D and 3D primitives available in ANSYS, dynamic analysis (harmonic and modal analysis), and thermal analysis.

CHAPTER 1

DYNAMIC ANALYSIS

Structural analysis is mainly concerned with finding out the behavior of a physical structure when subjected to force. This action can be in the form of load due to the weight of things such as people, furniture, wind, snow, and so on, or some other kind of excitation such as an earthquake, shaking of the ground due to a blast nearby, and so on. In essence all these loads are dynamic, including the self-weight of the structure because at some point in time these loads were not there. The distinction is made between the dynamic and the static analysis on the basis of whether the applied action has enough acceleration in comparison to the structure's natural frequency. If a load is applied sufficiently slowly, the inertia forces (Newton's first law of motion) can be ignored and the analysis can be simplified as static analysis. Structural dynamics, therefore, is a type of structural analysis that covers the behavior of structures subjected to dynamic (actions having high acceleration) loading. Dynamic loads include people, wind, waves, traffic, earthquakes, and blasts. Any structure can be subjected to dynamic loading. Dynamic analysis can be used to find dynamic displacements, time history, and modal analysis.

A dynamic analysis is also related to the inertia forces developed by a structure when it is excited by means of dynamic loads applied suddenly (e.g., wind blasts, explosion, and earthquake).

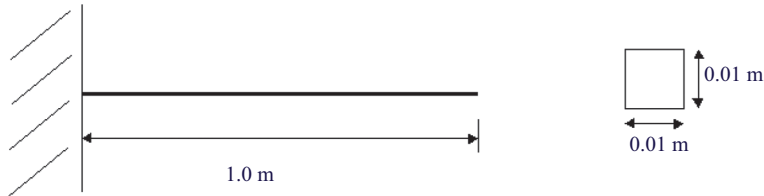
Dynamic analysis for simple structures can be carried out manually, but for complex structures finite element analysis can be used to calculate the mode shapes and frequencies.

1.1 TUTORIAL 1: HARMONIC ANALYSIS OF STRUCTURE

In this tutorial, the harmonic analysis of a cantilever beam will be addressed. Harmonic analysis is used to determine the response of a

2 • USING ANSYS FOR FINITE ELEMENT ANALYSIS

structure to harmonically time-varying loads. This tutorial was created using ANSYS 7.0. The purpose of this tutorial is to explain the steps required to perform harmonic analysis on the cantilever beam shown as follows.



Modulus of elasticity (E) = 206800(10⁶) N/m²

Density = 7830 kg/m³

We will now conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1–100 Hz. The following figure depicts the beam with the application of the load.



ANSYS provides three methods for conducting a harmonic analysis. These three methods are the Full, Reduced, and Modal Superposition methods. This example demonstrates the Full method because it is simple and easy to use as compared to the other two methods. However, this method makes use of the full stiffness and mass matrices and thus is the slower and costlier option.

1.1.1 STEP-BY-STEP ANSYS SOLUTION

1.1.1.1 Preprocessing: Defining the Problem

Student should be able to make simple cantilever model himself or otherwise use the following command list:

1.1.1.2 The Command Log File

```

/TITLE, Dynamic Analysis
/FILNAME,Dynamic,0          ! This sets the jobname to “Dynamic”
/PREP7
K,1,0,0
K,2,1,0
L,1,2
ET,1,BEAM3
R,1,0.0001,8.33e-10,0.01
MP,EX,1,2.068e11
MP,PRXY,1,0.33
MP,DENS,1,7830
LESIZE,ALL,,10
LMESH,1
FINISH

```

1.1.1.3 Solution: Assigning Loads and Solving

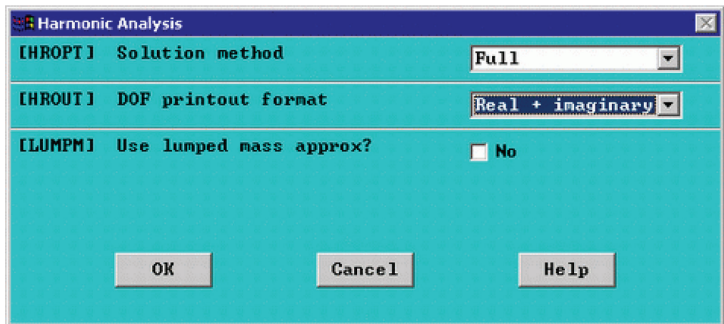
1. Define analysis type (Harmonic)
Solution > Analysis Type > New Analysis > Harmonic ANTYPE,3

2. Set options for analysis type

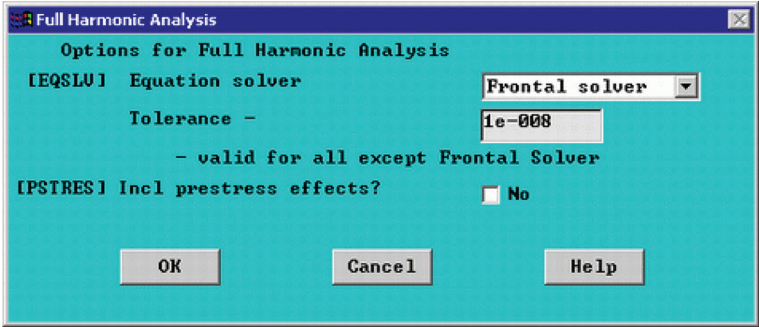
Select Solution > Analysis Type > Analysis Options.

The following window will appear.

As shown, select the Full Solution method, the Real + imaginary DOF (degrees of freedom) printout format and do not use lumped mass approx. Click “OK”



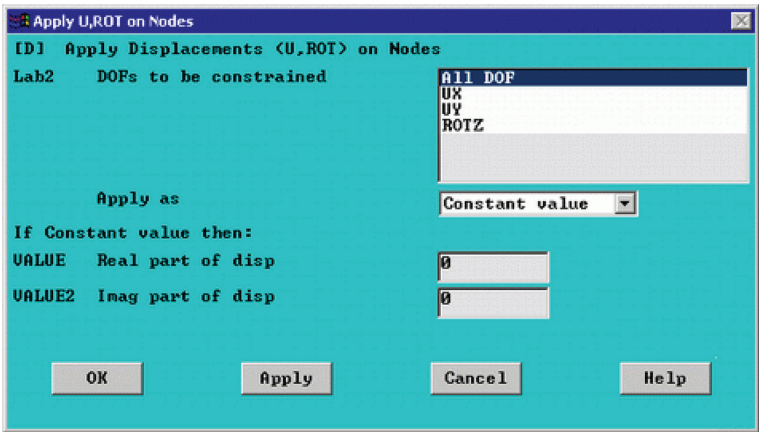
The following window will appear. Use the default settings (shown as follows).



3. Apply constraints

**Select Solution > Define Loads > Apply >
Structural > Displacement > On Nodes**

The following window will appear once you select the node at x=0 (Note small changes in the window compared to the static examples):



Constrain all DOF as shown in the above window.

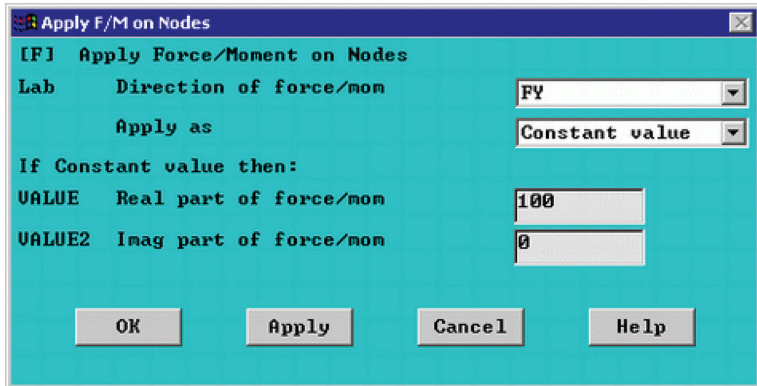
4. Apply Loads

**Select Solution > Define Loads > Apply > Structural >
Force/Moment > On Nodes**

Select the node at x=1 (far right). The following window will appear. Fill it in as shown to apply a load with a real value of 100 and an imaginary value of 0 in the positive “y” direction.

Note: By specifying a real and imaginary value of the load we are providing information on magnitude and phase of the load. In this case the magnitude of the load is 100 N and its phase is 0. Phase information is important when you have two or more cyclic loads

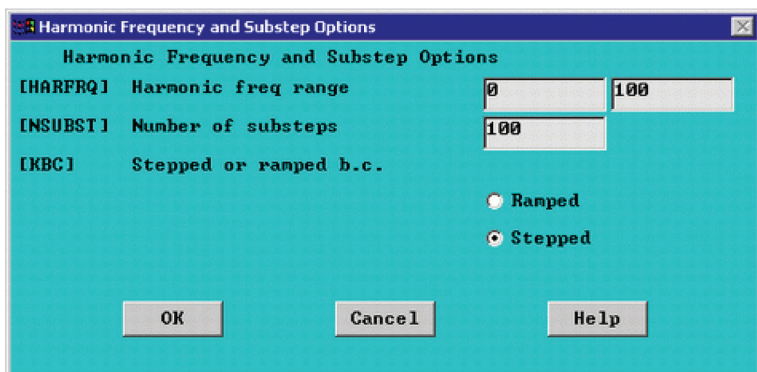
being applied to the structure as these loads could be in or out of phase. For harmonic analysis, all loads applied to a structure must have the SAME FREQUENCY.



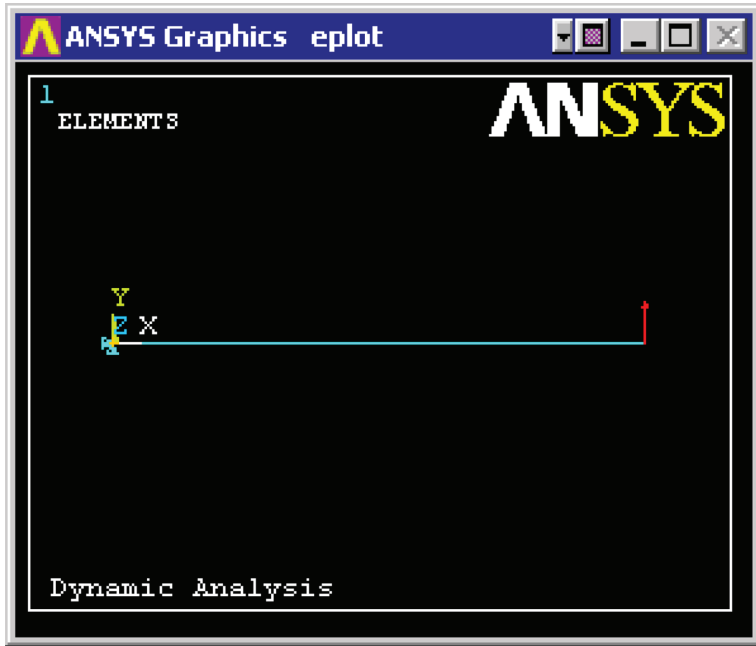
5. Set the frequency range

**Select Solution > Load Step Opts > Time/Frequency
> Freq and Substps...**

As shown in the following window, specify a frequency range of 0–100 Hz, 100 substeps, and stepped b.c. By doing this we will be subjecting the beam to loads at 1 Hz, 2 Hz, 3 Hz, 100 Hz. We will specify a stepped boundary condition (KBC) as this will ensure that the same amplitude (100 N) will be applied for each of the frequencies. The ramped option, on the other hand, would ramp up the amplitude where at 1 Hz the amplitude would be 1 N and at 100 Hz the amplitude would be 100 N.



You should now have the following in the ANSYS Graphics window



6. Solve the system

**Solution>Solve>CurrentLS
SOLVE**

1.1.1.4 Postprocessing: Viewing the Results

We want to observe the response at $x=1$ (where the load was applied) as a function of frequency. We cannot do this with General PostProcessing (POST 1); rather we must use TimeHist Post-Processing (POST26). POST26 is used to observe certain variables as a function of either time or frequency.

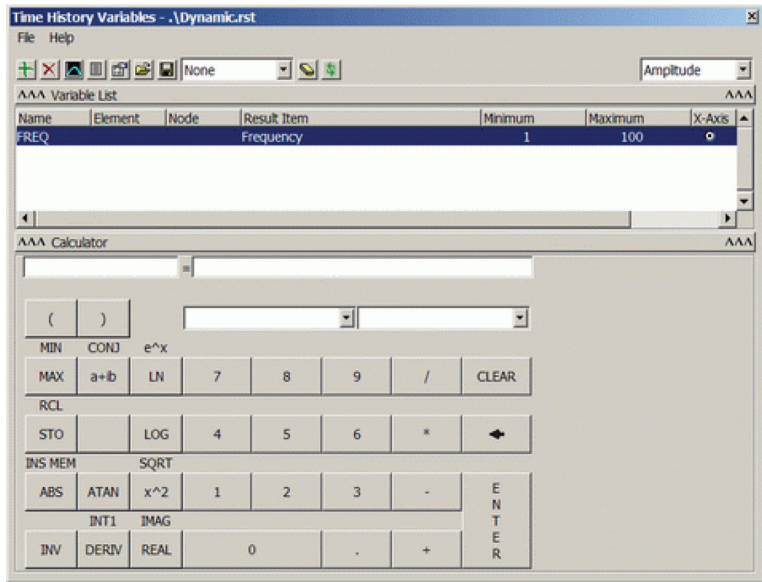
1. Open the TimeHist Processing (POST26) Menu
Select TimeHist Postpro from the ANSYS Main Menu.

2. Define variables

Here we have to define variables that we want to see plotted. By default, Variable 1 is assigned either Time or Frequency. In our case it is assigned Frequency. We want to see the displacement UY at the node at $x=1$, which is node #2. (To get a list of nodes and their attributes, select Utility Menu > List > nodes.)

Select Time HistPostpro> Variable Viewer...

And the following window should pop up:



Select Add (the green “+” sign in the upper left corner) from this window and the following window should appear:

