DYNAMIC ANALYSIS OF HYDRAULIC CYLINDER

by

Kutlay AKSÖZ

September, 2004
IZMIR
DYNAMIC ANALYSIS OF HYDRAULIC CYLINDER

A Thesis Submitted to the
Graduate School of Natural and Applied Sciences of
Dokuz Eylül University
In Partial Fulfillment of the Requirements for
the Degree of Master of Science in Mechanical Engineering,
Machine Theory and Dynamics Program

by
Kutlay AKSÖZ

September, 2004
IZMIR
We certify that we have read the thesis, entitled ‘Dynamic analysis of hydraulic cylinder’ completed by Kutlay AKSÖZ under supervision of Prof. Dr. Hira Karagülle and that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Supervisor

Committee Member

Committee Member

Approved by the
Graduate School of Natural and Applied Sciences

Prof.Dr.Cahit Helvaci
Director
ACKNOWLEDGEMENT

First of all, I would like to take the oppurtunity to thank my supervisor, Prof. Dr. Hira KARAGÜLLE, for his guidance and sharing of his knowledge, experience and time with me throughout this study.

I would also like to thank Assist. Prof. Zeki KIRAL for his helps with valuable suggestions and continuous moral support during this research.

I also extend my sincere thanks to Resrch. Assist. Levent MALGACA and Resrch. Assist. Gürkan SAKAR for their helps and supports.

Finally, I wish to express my sincere thanks to my family for their encouragement and endless patience.

Kutlay AKSÖZ, Izmir-2004
ABSTRACT

Many machines and machine mechanisms run under dynamic working conditions. The vibrations produced under dynamic conditions affect many important design parameters such as strength, production costs, productivity. In this thesis, the vibration analysis of a hydraulic cylinder subjected to dynamic loads is studied. Computer aided engineering (CAE) procedures are used to analyze the dynamic response of the cylinder walls. The finite element methods used in the analysis are applied by a computer aided design and analysis software ANSYS.

First of all, the vibration under the moving load problem is studied. The vibration of simply supported beam under a moving point load is analyzed by using ANSYS. The results are compared with the existing analytical solutions and the results of another CAE program I-DEAS in the literature. In addition to studies in the literature, it is considered that the moving load acts on to the beam periodically and dynamic results of these periodical loading are analyzed. According to this analysis, under the longer periodical loading conditions, the dynamic magnification becomes larger.

The studies on the moving load are then extended to the hydraulic cylinder. The finite element model of the cylinder is created. An ANSYS APDL code is developed to obtain the time-histories of the nodal excitation functions of the pressure loading created by the movement of the piston in the cylinder. APDL stands for ANSYS Parametric Design Language, a scripting language that one can use to automate the common tasks or even building the model in term of parameters. The working pressure and the piston velocity are considered in defining the loading functions. The
vibration analysis of the hydraulic cylinder is performed by using ANSYS after creating nodal excitation functions. The dynamic magnification values are calculated for various piston velocities. It is observed that the dynamic magnification value is dependent to piston velocity. Furthermore, the effects of damping in the dynamic analysis is investigated. As a result, it is observed that the finite element programs like ANSYS can be used to create the dynamic loading models of the hydraulic or pneumatic cylinders.

**Keywords:** Dynamic analysis, hydraulic cylinder, finite element method, ANSYS, dynamic magnification factor
ÖZET


Hareketli yük problemi daha sonra hidrolik silindirde ele alınmıştır. Hidrolik silindirin sonlu elemanlar modeli olusturulmuştur. Silindir içerisindeki piston hareketi sonucu oluşan basınç yüklemelerine ait zorlama kuvvetleri hazırlanan ANSYS APDL kodu ile olusturulmuştur. APDL, ANSYS programının ortak görevleri otomatiklestireni, ve hatta parametrik modellemeye imkan veren parametrik

Anahtar kelimeler: Dinamik analiz, hidrolik silindir, sonlu elemanlar yöntemi, ANSYS, dinamik yükseltme faktörü
CONTENTS

<table>
<thead>
<tr>
<th>Contents</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>List of Tables</td>
<td>VIII</td>
</tr>
<tr>
<td>List of Figures</td>
<td>IX</td>
</tr>
</tbody>
</table>

Chapter One
INTRODUCTION

1. Introduction                                1

Chapter Two
DYNAMIC ANALYSIS IN ANSYS

2.1 Introduction                               5
2.2 Dynamic Analysis Theory in ANSYS           6
   2.2.1 Definition of Transient Dynamic Analysis 6
   2.2.2 Solution Methods                        10
      2.2.2.1 The Full Method                     11
      2.2.2.2 The Reduced Method                  17
      2.2.2.3 The Mode Superposition Method       20
Chapter Three
DYNAMIC ANALYSIS OF BEAMS UNDER MOVING LOAD

3.1 Introduction .................................................................................. 23
3.2 Beam Subjected to Moving Load ..................................................... 24
3.3 Solution .......................................................................................... 25
  3.3.1 Selection of Time Intervals ......................................................... 26
  3.3.2 Excitation Functions ................................................................. 27
  3.3.3 Transient Analysis ................................................................. 27

Chapter Four
DYNAMIC ANALYSIS OF HYDRAULIC CYLINDER
SUBJECTED TO MOVING PRESSURE

4.1 Introduction .................................................................................. 36
4.2 Modeling of Hydraulic Cylinder ..................................................... 37
4.3 Static Analysis ............................................................................... 41
4.4 Vibration Analysis ........................................................................ 44
  4.4.1 Formation of Loading Functions ............................................... 44
  4.4.2 Modal Analysis ......................................................................... 44
  4.4.3 Transient Analysis ................................................................... 48
  4.4.4 The effect of damping in the transient dynamic analysis .......... 52

Chapter Five
CONCLUSIONS

Conclusions ......................................................................................... 53

References .......................................................................................... 55

Appendices .......................................................................................... 58
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Loads applicable in transient dynamic analysis</td>
<td>14</td>
</tr>
<tr>
<td>2.2</td>
<td>Load step options for transient dynamic analysis</td>
<td>14</td>
</tr>
<tr>
<td>3.1</td>
<td>Natural frequencies of the beam</td>
<td>26</td>
</tr>
<tr>
<td>4.1</td>
<td>Material properties of cylinder</td>
<td>38</td>
</tr>
<tr>
<td>4.2</td>
<td>Static displacement values of nodes 1522 and 2809 (m)</td>
<td>42</td>
</tr>
<tr>
<td>4.3</td>
<td>Natural Frequencies of Cylinder, Bc-type1</td>
<td>45</td>
</tr>
<tr>
<td>4.4</td>
<td>Natural Frequencies of Cylinder, Bc-type2</td>
<td>45</td>
</tr>
<tr>
<td>4.5</td>
<td>The maximum displacement values for different damping coefficients</td>
<td>52</td>
</tr>
</tbody>
</table>
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 2.1</td>
<td>Examples of load-versus-time curves</td>
<td>11</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Finite element model of the beam</td>
<td>24</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>Static loading model of the beam</td>
<td>25</td>
</tr>
<tr>
<td>Figure 3.3</td>
<td>Static deflection of the beam</td>
<td>25</td>
</tr>
<tr>
<td>Figure 3.4</td>
<td>Traveling load time history for the $i^{th}$ node</td>
<td>28</td>
</tr>
<tr>
<td>Figure 3.5</td>
<td>Cycle definition</td>
<td>29</td>
</tr>
<tr>
<td>Figure 3.6</td>
<td>Time history for normalized mid-span displacement obtained by ANSYS, $\alpha_p=0.25$, single cycle</td>
<td>30</td>
</tr>
<tr>
<td>Figure 3.7</td>
<td>Dynamic magnification factor-versus- $\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale, single cycle</td>
<td>30</td>
</tr>
<tr>
<td>Figure 3.8</td>
<td>Normalized time for the maximum mid-span displacements-versus- $\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1 , single cycle</td>
<td>31</td>
</tr>
<tr>
<td>Figure 3.9</td>
<td>Time history for normalized mid-span displacement obtained by ANSYS, $\alpha_p=0.25$, double cycle</td>
<td>32</td>
</tr>
<tr>
<td>Figure 3.10</td>
<td>Dynamic magnification factor versus $\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1 , double cycle</td>
<td>33</td>
</tr>
<tr>
<td>Figure 3.11</td>
<td>Normalized time for the maximum mid-span displacements-versus- $\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1 , double cycle</td>
<td>33</td>
</tr>
</tbody>
</table>
Figure 3.12 Time history of normalized mid-span displacement for $\alpha_p = 0.25$...

Figure 3.13 Normalized time for the maximum mid-span displacements-versus-$\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1, double cycle, corrected...

Figure 4.1 The finite element model and the cylinder dimensions...

Figure 4.2 Boundary condition a) Bc-type 1, b) Bc-type 2.......

Figure 4.3 Cylinder filled with 250 bar pressure, a) isometric view, b) side views...

Figure 4.4 Static displacement distribution of cylinder, Bc-type 1...

Figure 4.5 Static displacement distribution of cylinder, Bc-type 2...

Figure 4.6 Mode Shape Example, Bc-type 1, Mode 3 (576.85Hz)....

Figure 4.7 Mode Shape Example, Bc-type 1, Mode 9 (1060.2Hz)....

Figure 4.8 Mode Shape Example, Bc-type 2, Mode 1 (623.71Hz)....

Figure 4.9 Mode Shape Example, Bc-type 2, Mode 5 (1139.2Hz)....

Figure 4.10 Time history for normalized displacements of node 1522, Bc-type 1...

Figure 4.11 Time history for normalized displacements of node 1522, Bc-type 2...

Figure 4.12 Time history for normalized displacements of node 2809, Bc-type 1...

Figure 4.13 Time history for normalized displacements of node 2809, Bc-type 2...

Figure 4.14 Normalized time for the maximum mid-stroke displacements-versus-frequency, Bc-type 1...
Hydraulic systems are widely used systems in industry. While they are used widely in the industry, the system components like pumps, valves, cylinders are always became investigation topics in the history. Hydraulic cylinders are one of the most common components of the hydraulic systems used in many engineering applications like; automatic manufacturing and montage lines, heavy construction equipments, control systems, sensitive measurement and test systems. They are used for producing linear motion in the hydraulic systems and they convert hydraulic energy to mechanical energy.

One of the most important factors considering at the design step of these equipments is working conditions of cylinder. Cylinders have different working frequencies according to their usage fields. While the huge sized cylinders used in systems that requires higher force and power inputs, work generally in lower frequencies, the small sized cylinders used in sensitive application fields like test and measurement systems can have higher working frequencies.

At the lower working frequency situations, pressure effect on the cylinder is considered as static load, and the hydraulic system equipments are designed according to this criterion. Besides this, at the design procedure of cylinders with higher working frequencies, the dynamic effect with respect to instantaneous change of pressure must be taken into consideration as well as the static analysis. There are some studies in the literature that interest with the dynamic response of cylinders. Tzeng & Hopkins (1996) study the dynamic response of composite cylinders.
subjected to a moving internal pressure. Tzeng (1998) also analyzes the resonance of stress waves in the cylinder at the instant and location of the pressure front passage, when the velocity of the moving load approaches to the critical velocity. De Faria (2004) investigates the vibration of a cylinder panel with a moving force or mass by using finite element methods. He analyzes the effects of the panel curvature, moving load velocity and the moving mass to main structure ratio on the dynamic response. He observes that the dynamic response of the cylinder is significantly affected by rapid traversing velocities or heavy moving masses. Beltman & Shepherd (2002) suggest that the flexural waves can result in much higher strain and stresses than static loading with the same loading pressure. The wave propagation in a hollow cylinder is analyzed for pressure and velocity prescribed at its inner boundary (El Rahep, 2004).

The dynamic loading of cylinder can be classified as a moving load problem in which various parameters play a role. The moving load problem is studied by engineers and researchers for many years and is still an interesting engineering subject. The studies on the moving load problem are generally focused on the moving load on beams which are elastically supported or not supported.

A basic study upon the moving load problem and reference data is given by Olsson (1991). Olsson studies the dynamics of the beam subjected to a constant force moving at a constant speed and presents analytical and finite element solutions. Thambiratnam & Zhuge (1996) study the dynamics of beams on an elastic foundation and subjected to moving loads by using the finite element method. They investigated the effect of the foundation stiffness, traveling speed and the span length of the beam on the dynamic magnification factor which is defined as the ratio of the maximum displacement in the time history of the mid-span to the static mid-span displacement. Recently, Gutierrez & Laura (1997) consider the beam with non-uniform cross-section and present the analytical and approximate solutions for different boundary conditions. Wang (1997) analyzes the multi-span Timoshenko beams subjected to a concentrated moving force by using the mode superposition method and made a comparison between the Bernoulli-Euler beam and Timoshenko beam. Wu & Shih
(2000) study the dynamic responses of railway and carriage under the high-speed moving loads and consider the action of the multi-roller carriage. They employ the finite element method in the solution and investigate the influence of the total number of carriage and the spacing between the rollers on the dynamic response of the railway. Rao (2000) investigates the inertial effects of the moving load by using the mode superposition method and makes a comparison between the moving load and moving mass models. Recently experimental studies on the moving force identification are carried out by Law et al. (1999), Law & Zhu (2000) and Chan et al. (2000). These studies are addressed to the inverse problem in which the forces acting on the structure are identified by using the vibration responses without the knowledge of the vehicle characteristics.

This thesis documents the approach used in pursuing the research objectives, together with the results obtained. The thesis is organized as follows:

Chapter 1 is an introductory chapter and includes the literature review on the moving load problem in general and dynamic responses of cylinders. The objectives of this thesis and the method are explained.

Chapter 2 details the theory of dynamic analysis used in the CAE program ANSYS.

Chapter 3 presents the moving load problem on beams which forms the basis of the hydraulic cylinder dynamic analysis. The vibration response of a simple supported beam with and without elastic foundation to a moving single point load is analyzed performing the finite element vibration analysis. The dynamic magnification value versus the load traveling speed obtained and compared with the results of the similar studies in the literature.

Chapter 4 provides the details of the dynamic modeling of the hydraulic cylinder. The geometry and the finite element model of the cylinder are given. The method for generating the nodal excitation functions is described. The finite element vibration analysis and selection of the time step is described. Dynamic
magnification values obtained under the action of pressure loading are presented for the hydraulic cylinder.

Chapter 5 draws general and specific conclusions from the research work documented in this thesis.

This thesis includes two appendices. Appendix 1 presents the written ANSYS codes used for the analysis of the beam subjected to moving singular force. Appendix 2 presents the written ANSYS codes used for the analysis of the hydraulic cylinder under moving internal pressure.
CHAPTER TWO

DYNAMIC ANALYSIS IN ANSYS

2.1 Introduction

The finite element method is a numerical procedure that can be applied to obtain solutions to variety of problems in engineering. Steady, transient, linear, or nonlinear problems in stress analysis, heat transfer, fluid flow, and electromagnetism problems may be analyzed with finite element methods. The origin of the modern finite element method may be traced back to early 1900s when some investigators approximated and modeled elastic continua using discrete equivalent elastic bars. However, Courant (1943) has been credited with being the first person to develop the finite element method. In a paper published in the early 1940s, Courant used piecewise polynomial interpolation over triangular subregions to investigate torsion problems.

The next significant step in the utilization of the finite element methods was taken by Boeing in the 1950s when Boeing, followed by others, used triangular stress elements to model airplane wings. Yet it was not until 1960 that Clough (1975) made the term “finite element” popular. During the 1960s, investigators began to apply the finite element method to other areas of engineering, such as heat transfer and seepage flow problems. Zienkiewicz & Cheung (1967) wrote the first book entirely devoted to the finite element method in 1967. In 1971, ANSYS was released for the first time.
ANSYS is a comprehensive general-purpose finite element computer program that contains over 100,000 lines of codes. ANSYS is capable of performing static, dynamic, heat transfer, fluid flow, and electromagnetism analyses. ANSYS has been leading FEA program for well over 20 years. The current version of ANSYS has been completely new look, with multiple windows incorporating a graphic user interface (GUI), pulldown menus, dialog boxes, and a tool bar. Today, ANSYS can be found in use in many engineering fields, including aerospace, automotive, electronics, and nuclear.

2.2 Dynamic Analysis Theory In ANSYS

2.2.1 Definition of Transient Dynamic Analysis

Transient dynamic analysis (sometimes called time-history analysis) is a technique used to determine the dynamic response of a structure under the action of any general time-dependent loads. This type of analysis can be used to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to any combination of static, transient, and harmonic loads. The time scale of the loading is such that the inertia or damping effects are considered to be important. If the inertia and damping effects are not important, it might be able to use a static analysis instead.

The transient analysis solution method used depends on the DOFs involved. Structural, acoustic, and other second order systems (that is, the systems are second order in time) are solved using one method and the thermal, magnetic, electrical and other first order systems are solved using another. Each method is described subsequently. If the analysis contains both first and second order DOFs (e.g. structural and magnetic), then each DOF is solved using the appropriate method. For matrix coupling between first and second order effects such as for piezoelectric analysis, a combined procedure is used.
The transient dynamic equilibrium equation of interest is as follows for a linear structure:

\[
[M][\ddot{u}]+[C][\dot{u}]+[K][u] = \{F^a\}
\]  

(2.1)

where:

\begin{align*}
[M] & = \text{structural mass matrix} \\
[C] & = \text{structural damping matrix} \\
[K] & = \text{structural stiffness matrix} \\
{\ddot{u}} & = \text{nodal acceleration vector} \\
{\dot{u}} & = \text{nodal velocity vector} \\
{u} & = \text{nodal displacement vector} \\
{F^a} & = \text{applied load vector}
\end{align*}

There are two methods in the ANSYS program which can be employed for the solution of the linear equation (2.1): the forward difference time integration method and the Newmark time integration method. The forward difference method is used for explicit transient analyses.

The Newmark method is used for implicit transient analyses and is described below,

The Newmark method uses finite difference expansions in the time interval $\Delta t$, in which it is assumed that (Bathe, 1982):

\[
{\ddot{u}}_{n+1} = {\dot{u}}_{n+1} + [(1-\delta)\dot{u}_n + \delta \ddot{u}_n] \Delta t
\]  

(2.2)

\[
{u}_{n+1} = {u}_n + [{\dot{u}}_n] \Delta t + \left[\frac{1}{2} - \alpha \right] {\ddot{u}}_n + \alpha {\ddot{u}}_{n+1} \right] \Delta t^2
\]  

(2.3)

where:

\[
\alpha, \delta = \text{Newmark integration parameters}
\]
\[ \Delta t = t_{n+1} - t_n \]

\[ \{u_n\} = \text{nodal displacement vector at time } t_n \]

\[ \{\dot{u}_n\} = \text{nodal velocity vector at time } t_n \]

\[ \{\ddot{u}_n\} = \text{nodal acceleration vector at time } t_n \]

\[ \{u_{n+1}\} = \text{nodal displacement vector at time } t_{n+1} \]

\[ \{\dot{u}_{n+1}\} = \text{nodal velocity vector at time } t_{n+1} \]

\[ \{\ddot{u}_{n+1}\} = \text{nodal acceleration vector at time } t_{n+1} \]

Since the primary aim is the computation of displacements \( \{u_{n+1}\} \), the governing equation (2.1) is evaluated at time \( t_{n+1} \) as:

\[ [M]\ddot{u}_{n+1} + [C]\dot{u}_{n+1} + [K]u_{n+1} = \{F^*\} \quad (2.4) \]

The solution for the displacement at time \( t_{n+1} \) is obtained by first rearranging equations (2.2) and (2.3), such that:

\[ \{\ddot{u}_{n+1}\} = a_0 (\{u_{n+1}\} - \{u_n\}) - a_2 \{\dot{u}_n\} - a_3 \{\ddot{u}_n\} \quad (2.5) \]

\[ \{\dot{u}_{n+1}\} = \{\dot{u}_n\} + a_6 \{\ddot{u}_n\} + a_7 \{\ddot{u}_{n+1}\} \quad (2.6) \]

where:

\[ a_0 = \frac{1}{\alpha\Delta t^2} \quad a_4 = \frac{\delta}{\alpha} - 1 \]

\[ a_1 = \frac{\delta}{\alpha\Delta t} \quad a_5 = \frac{\Delta t}{2} \left( \frac{\delta}{\alpha} - 2 \right) \]

\[ a_2 = \frac{1}{\alpha\Delta t} \quad a_6 = \Delta t(1 - \delta) \]
\[ a_3 = \frac{1}{2\alpha} - 1 \quad a_\gamma = \delta \Delta t \]

Noting that \( \{ \ddot{u}_{n+1} \} \) in equation (2.5) can be substituted into equation (2.6), equations for \( \{ \ddot{u}_{n+1} \} \) and \( \{ u_{n+1} \} \) can be expressed only in terms of the unknown \( \{ u_{n+1} \} \), The equations for \( \{ \ddot{u}_{n+1} \} \) and \( \{ u_{n+1} \} \) are then combined with equation (2.4) to form:

\[
\left( [M] + a_1 \{ C \} + [M] \right) \{ u_{n+1} \} = \{ F^* \} +
\left( [M]a_0 \{ \dot{u}_n \} + a_3 \{ \ddot{u}_n \} \right) + \left[ C \right]a_1 \{ u_n \} + a_4 \{ \dot{u}_n \} + a_5 \{ \ddot{u}_n \}
\]

\[ (2.7) \]

Once a solution is obtained for \( \{ u_{n+1} \} \), velocities and accelerations are updated as described in equations (2.5) and (2.6).

As described by Zienkiewicz (1977), the solution of equation (2.4) by means of Newmark equations (2.2) and (2.3) is unconditionally stable for:

\[
\alpha \geq \frac{1}{4} \left( \frac{1}{2} + \delta \right)^2 , \quad \delta \geq \frac{1}{2} , \quad \frac{1}{2} + \delta + \alpha > 0
\]

\[ (2.8) \]

In the ANSYS implementation, the selected values for the Newmark parameters are as follows;

\[
\alpha = \frac{1}{4} (1 + \gamma)^2 , \quad \delta = \frac{1}{2} + \gamma
\]

\[ (2.9) \]

where: \( \gamma \) = amplitude decay factor.

By inspection of equations (2.8) and (2.9), unconditional stability is achieved when \( \delta = \frac{1}{2} + \gamma , \alpha \geq \frac{1}{4} \left( \frac{1}{2} + \delta \right)^2 \), and \( \gamma \geq 0 \). Thus all solutions of equation (2.4) are stable if \( \gamma \geq 0 \).
Typically the amplitude decay factor in equation (2.9) takes a small value (the default is 0.005). The Newmark method becomes the constant average acceleration method when $\gamma = 0$, which in turns means $\alpha = \frac{1}{4}$ and $\delta = \frac{1}{2}$ (Bathe, 1982). Results from the constant average acceleration method do not show any numerical damping in terms of displacement amplitude errors. If other sources of damping are not present, the lack of numerical damping can be undesirable in that the higher frequencies of the structure can produce unacceptable levels of numerical noise (Zienkiewicz, 1977). A certain level of numerical damping is usually desired and is achieved by degrading the Newmark approximation by setting $\gamma > 0$.

2.2.2 Solution Methods

There are three methods available to do a transient dynamic analysis: full, reduced, and mode superposition.

2.2.2.1 The Full Method

The full method uses the full system matrices to calculate the transient response (no matrix reduction). It is the most powerful of the three methods because it allows all types of nonlinearities to be included (plasticity, large deflections, large strain, etc). The advantages of the full method are:

- It is easy to use, because there is no need to choose master of freedom or mode shapes.
- It allows all types of nonlinearities.
- It uses full matrices, so no mass matrix approximation is involved.
- All displacements and stresses are calculated in a single pass.
- It accepts all types of loads: nodal forces, imposed (non-zero) displacements (although not recommended), and element loads (pressures and temperatures).
• It allows effective use of solid-model loads.

The main disadvantage of the full method is that it is more expensive than either of the other methods.

The full solution method solves equation (2.7) directly and makes no additional assumptions. In a nonlinear analysis, the Newton-Raphson method is employed along with the Newmark assumptions. The inversion of equation (2.7) (or its nonlinear equivalent) employs the same wavefront solver used for a static analysis.

Inherent to the Newmark method is that the values of \( \{u_0\}, \{\dot{u}_0\} \) and \( \{\ddot{u}_0\} \) at the start of the transient must be known. Non-zero initial conditions are input by performing a static analysis load step (or load steps) prior to the start of the transient itself. Static load steps are performed in a transient analysis by turning off the transient time integration effects. The initial conditions are outlined in the subsequent paragraphs. Cases referring to "no previous load step" mean that the first load step is transient.

A transient analysis, by definition, involves loads that are functions of time. To specify such loads, it is needed to divide the load-versus-time curve into suitable load steps. Each "corner" on the load-time curve may be one load step, as shown in Figure 2-1.

![Figure 2.1 Examples of load-versus-time curves](image-url)
The first load step applied is usually to establish initial conditions. Then the loads and load step options are applied for the subsequent transient load steps. For each load step, it is needed to specify both load values and time values, along with other load step options such as whether to step or ramp the loads, automatic time stepping, etc. Then each load step is written to a file and all load steps are solved together.

The first step in applying transient loads is to establish initial conditions (that is, the condition at $t = 0$). A transient dynamic analysis requires two sets of initial conditions (because the equations being solved are of second order): initial displacement ($u_0$) and initial velocity ($\dot{u}_0$). If no special action is taken, both ($u_0$) and ($\dot{u}_0$) are assumed to be zero. Initial accelerations ($\ddot{u}_0$) are always assumed to be zero, but you can specify nonzero initial accelerations by applying appropriate acceleration loads over a small time interval.

The initial displacements are:

$$\{u_0\} = \begin{cases} \{0\} & \text{if no previous load step available and no IC command is used} \\ \{\dot{u}\} & \text{if no previous load step available but IC command(s) is used} \\ \{u_s\} & \text{if previous load step available} \end{cases}$$

where: $\{u_0\} = \text{vector of initial displacements}$
$\{\dot{u}\} = \text{displacement vector specified by the IC command}$
$\{u_s\} = \text{displacement vector resulting from a static analysis of the previous load step}$

The initial velocities are:

$$\{\dot{u}_0\} = \begin{cases} \{0\} & \text{if no previous load step available and no IC command is used} \\ \{\dot{u}\} & \text{if no previous load step available but IC command(s) is used} \\ \{u_s\} - \frac{u_{s-1}}{\Delta} & \text{if previous load step available} \end{cases}$$
where:

\[ \{ \dot{u}_0 \} = \text{vector of initial velocities} \]

\[ \{ \dot{u}_s \} = \text{velocity vectors specified by the IC command} \]

\[ \{ \dot{u}_r \} = \text{velocity vector resulting from a static analysis of the previous load step} \]

\[ \{ u_{-1} \} = \text{displacement corresponding to the point before solution.} \]

\[ \{ u_{s-1} \} \text{is } \{ 0 \} \text{if } \{ u_s \} \text{is the first solution of the analysis (i.e. load step 1 substep 1).} \]

\[ \Delta t = \text{time increment between } s \text{ and } s-1 \]

The initial acceleration is simply:

\[ \{ \ddot{u}_0 \} = \{ 0 \} \]

where:

\[ \{ \ddot{u}_0 \} = \text{vector of initial accelerations} \]

If a nonzero initial acceleration is required as for a free fall problem, an extra load step at the beginning of the transient can be used. This load step would have a small time span, step boundary conditions, and a few time steps which would allow the acceleration to be well represented at the end of the load step.

Table 2.1 summarizes the loads applicable to a transient dynamic analysis. Except for inertia loads, you can define loads either on the solid model (keypoints, lines, and areas) or on the finite element model (nodes and elements).
Table 2.1  Loads applicable in transient dynamic analysis

<table>
<thead>
<tr>
<th>Load Type</th>
<th>Category</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement (UX, UY, LIZ, ROTX, ROTY, ROTZ)</td>
<td>Constraints</td>
</tr>
<tr>
<td>Force, Moment (FX, FY, FZ, MX, MY, MZ)</td>
<td>Forces</td>
</tr>
<tr>
<td>Pressure (PRES)</td>
<td>Surface Loads</td>
</tr>
<tr>
<td>Temperature (TEMP)</td>
<td>Body Loads</td>
</tr>
<tr>
<td>Fluence (FLUE)</td>
<td></td>
</tr>
<tr>
<td>Gravity, Spinning, etc.</td>
<td>Inertia Loads</td>
</tr>
</tbody>
</table>

The following load step options are available for a transient dynamic analysis:

Table 2.2  Load step options for transient dynamic analysis

<table>
<thead>
<tr>
<th>Options</th>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Dynamic Options</strong></td>
<td></td>
</tr>
<tr>
<td>Time Integration Effects</td>
<td>TIMINT</td>
</tr>
<tr>
<td>Transient Integration Parameters</td>
<td>TINTP</td>
</tr>
<tr>
<td>Damping</td>
<td>ALPHAD, BETAD, MP, DAMP</td>
</tr>
<tr>
<td><strong>General Options</strong></td>
<td></td>
</tr>
<tr>
<td>Time</td>
<td>TIME</td>
</tr>
<tr>
<td>Stepped or Ramped Loads</td>
<td>KBC</td>
</tr>
<tr>
<td>Integration Time Step</td>
<td>NSUBST, DELTIM</td>
</tr>
<tr>
<td>Automatic Time Stepping</td>
<td>AUTOTS</td>
</tr>
<tr>
<td><strong>Output Control Options</strong></td>
<td></td>
</tr>
<tr>
<td>Printed Output</td>
<td>OUTPR</td>
</tr>
<tr>
<td>Database and Results File Output</td>
<td>OUTRES</td>
</tr>
<tr>
<td>Extrapolation of Results</td>
<td>ERESX</td>
</tr>
</tbody>
</table>
Dynamic options include the following:

- **Time Integration Effects [TIMINT]**
  Time integration effects must be turned on for inertia and damping effects to be included in the analysis (otherwise a static solution is performed). The default is to include time integration effects. This option is useful to begin a transient from an initial static solution; that is, the first load steps are solved with the time integration effects off.

- **Transient Integration Parameters [TINTP]**
  Transient integration parameters control the nature of the Newmark time integration technique. The default is to use the constant average acceleration scheme.

- **Damping**
  Damping in some form is present in most structures and should be included in your analysis. You can specify three forms of damping in a transient dynamic analysis:
  - Alpha (mass) damping [ALPHAD]
  - Beta (stiffness) damping [BETAD]
  - Material-dependent beta damping [MP,DAMP]
  All three forms result in frequency-dependent damping ratio.

General options include the following:

- **Time [TIME]**
  This option specifies time at the end of the load step.

- **Stepped or Ramped Loads [KBC]**
  This option indicates whether to ramp the load change over the load step [KBC] or to step-apply the load change [KBC,1]. The default is ramped.
- Integration Time Step [DELTIM or NSUBST]
  The integration time step is the time increment used in the time integration of the equations of motion, it can be specified directly [DELTIM] or indirectly, in terms of the number of substeps [NSUBST]. The time step size determines the accuracy of the solution: the smaller its value, the higher the accuracy. Several factors should be considered in order to calculate a "good" integration time step.

- Automatic Time Stepping [AUTOTS]
  This option, also known as time-step optimization in a transient analysis, increases or decreases the integration time step based on the response of the structure. For most problems, we recommend that you turn on automatic time stepping, with upper and lower limits for the integration time step. These limits, specified using DELTIM or NSUBST, help to limit the range of variation of the time step.

Inertia, damping and static loads on the nodes of each element are computed. The inertial load part of the element output is computed by:

\[
\{F_\text{e}^m\} = [M_\text{e}] [\ddot{u}_\text{e}]
\]

where:
- \(\{F_\text{e}^m\}\) = vector of element inertial forces
- \([M_\text{e}]\) = element mass matrix
- \(\{\ddot{u}_\text{e}\}\) = element acceleration vector

The acceleration of a typical DOF is given by equation (2.5) for time \(t_{n+1}\). The acceleration vector \(\{\ddot{u}_\text{e}\}\) is the average acceleration between time \(t_{n+1}\) and time \(t_n\), since the Newmark assumptions (equations (2.2) and (2.3)) assume the average acceleration represents the true acceleration.
The damping load part of the element output is computed by:

\[
\{F^d_e\} = [C_e]\{\dot{u}_e\}
\]  \hfill (2.14)

where:
- \(\{F^d_e\}\) = vector of element damping forces
- \([C_e]\) = element damping matrix
- \(\{\dot{u}_e\}\) = element velocity vector

The static load is part of the element output computed in the same way as in a static analysis. The nodal reaction loads are computed as the negative of the sum of all three types of loads (inertia, damping, and static) over all elements connected to a given fixed displacement node.

2.2.2.2 The Reduced Method

The reduced method condenses the problem size by using master degrees of freedom and reduced matrices. After the displacements at the master DOF have been calculated, ANSYS expands the solution to the original full DOF set. The advantages of the reduced method are:

- It is faster and less expensive than the full method.

The disadvantages of the reduced method are:

- The initial solution calculates only the displacements at the master DOF. A second step, known as the expansion pass, is required for a complete displacement, stress and force solution. (However, the expansion pass might not be needed for some applications.)

- Element loads (pressures, temperatures, etc) cannot be applied. Accelerations, however, are allowed.

- All loads must be applied at user-defined master degrees of freedom. (This limits the use of solid-model loads)

- The time step must remain constant throughout the transient, so automatic time-stepping is not allowed.
• The only nonlinearity allowed is simple node-to-node contact (gap condition).

The reduced solution method uses reduced structure matrices to solve the time-dependent equation of motion (equation (2.1)) for linear structures. The solution method imposes the following additional assumptions and restrictions:

• Constant [M], [C], and [K] matrices, (A gap condition is permitted as described below.). This implies no large deflections or change of stress stiffening, as well as no plasticity, creep, or swelling.
• Constant time step size.
• No element load vectors. This implies no pressures or thermal strains. Only nodal forces applied directly at master DOF or acceleration effects acting on the reduced mass matrix are permitted.
• Non-zero displacements may be applied only at master DOF.

This method usually runs faster than the full transient dynamic analysis by several orders of magnitude, principally because the matrix on the left-hand side of equation (2.7) needs to be inverted only once and the transient analysis is then reduced to a series of matrix multiplications. Also, the technique of "matrix reduction", is used in this method, so that the matrix representing the system will be reduced to the essential DOFs required to characterize the response of the system. These essential DOFs are referred to as the "master degrees of freedom". The reduction of equation (2.7) for the reduced transient method results in:

\[
\begin{align*}
\{a_0 \hat{\mathbf{M}} + a_1 \hat{\mathbf{C}} + a_2 \hat{\mathbf{K}}\} \{\hat{\mathbf{u}}_{n+1}\} &= \{\hat{\mathbf{F}}\} + \\
\hat{\mathbf{M}}\left\{a_0 \{\hat{\mathbf{u}}_n\} + a_2 \{\hat{\mathbf{u}}_n\} + a_3 \{\hat{\mathbf{u}}_n\}\right\} + \hat{\mathbf{C}}\left\{a_1 \{\hat{\mathbf{u}}_n\} + a_4 \{\hat{\mathbf{u}}_n\} + a_5 \{\hat{\mathbf{u}}_n\}\right\}
\end{align*}
\]  

(2.15)

where the coefficients \(a_i\) are defined after equation (2.6). The ^ symbol is used to denote reduced matrices and vectors. [\hat{\mathbf{K}}] may contain prestressed effects corresponding to a non-varying stress. These equations, which have been reduced to
the master DOFs, are then solved by inverting the left-hand side of equation (2.15) and performing a matrix multiplication at each time step.

For the initial conditions, a static solution is done at time = 0 using the given loads to define \( \{\hat{u}_0\}, \{\hat{\dot{u}}_0\} \) and \( \{\hat{\ddot{u}}_0\} \) are assumed to be zero.

A "quasi-linear" analysis variation is also available with the reduced method. This variation allows interfaces (gaps) between any of the master DOFs and ground, or between any pair of master DOFs. If the gap is initially closed, these interfaces are accounted for by including the stiffness of the interface in the stiffness matrix, but if the gap should later open, a force is applied in the load vector to nullify the effect to the stiffness. If the gap is initially open, it causes no effect on the initial solution, but if it should later close, a force is again applied in the load vector.

The force associated with the gap is:

\[
F_{sp} = k_{sp} u_g
\]

(2.16)

where: 
- \( F_{sp} \) = gap stiffness
- \( u_g = u_A - u_B - u^{sp} \)
- \( u^A, u^B \) = displacement across gap (must be master degrees of freedom)
- \( u^{sp} \) = initial size of gap

This procedure adds an explicit term to the implicit integration procedure. An alternate procedure is to use the full method, modeling the linear portions of the structure as superelements and the gaps as gap elements. This latter procedure (implicit integration) normally allows larger time steps because it modifies both the stiffness matrix and load vector when the gaps change status.

The expansion pass of the reduced transient analysis involves computing the displacements at slave DOFs (see equation (2.16)) and computing element stresses.
The reaction load values represent the negative of the sum of the above static loads over all elements connected to a given fixed displacement node. Damping and inertia forces are not included in the reaction loads.

2.2.2.3 The Mode Superposition Method

The mode superposition method sums factored mode shapes (eigenvectors) from a modal analysis to calculate the structure's response. This is the only method available in the ANSYS/LinearPlus program. Its advantages are:

- It is faster and less expensive than the reduced or the full method for many problems.
- Element loads applied in the preceding modal analysis can be applied in the transient dynamic analysis.
- It accepts modal damping (damping ratio as a function of mode number).

The disadvantages of the mode superposition method are:

- The time step must remain constant throughout the transient, so automatic time-stepping is not allowed.
- The only nonlinearity allowed is simple node-to-node contact (gap condition).
- It should not be used for a "floating" or disjoint structure.
- When using PowerDynamics, initial conditions may not have previously-applied loads or displacements.
- It does not accept imposed (non-zero) displacements.

The mode superposition method uses the natural frequencies and mode shapes of a linear structure to predict the response to transient forcing functions. This solution method imposes the following additional assumptions and restrictions:
• Constant \([K]\) and \([M]\) matrices. (A gap condition is permitted as described under the reduced solution method.) This implies no large deflections or change of stress stiffening, as well as no plasticity, creep, or swelling.
• Constant time step size.
• There are no element damping matrices, however, various types of system damping are available.
• Time varying imposed displacements are not allowed.

The initial values of the displacements \(\{u\}\) come from a static analysis (equation (2.1)) of

\[
[K][u_o] = \{F_0\} \quad (2.17)
\]

where: \(\{F_0\} = \) the forces applied at time = 0

so that \(\{u_o\} = [K]^{-1}\{F_0\}\) \( (2.18)\)

The initial values of the modal coordinates are computed from \(\{u_o\}\) by

\[
\{y_o\} = [\Phi]^{-1}\{u_o\} \quad (2.19)
\]

where: \([\Phi]\) = is the matrix of mode shape vectors
The load vector, which must be converted to modal coordinates at each time step, is given by,

\[
\{F\} = \{F^{nd}\} + s\{F^{s}\} + \{F_{gp}\} + \{F_{mu}\}
\]  \hspace{1cm} (2.20)

where:

- \(\{F^{nd}\}\) = nodal force vector
- \(s\) = load vector scale factor
- \(\{F^{s}\}\) = load vector from the modal analysis.
- \(\{F_{gp}\}\) = gap force vector (equation (2.16)).
- \(\{F_{mu}\}\) = inertial force \((\{F_{mu}\} = [M]\{a\})\)
- \(\{a\}\) = acceleration vector

If the modal analysis was performed using the reduced method, then the matrices and vectors in the above equations would be in terms of the master DOFs (e.g. \(\{\hat{u}\}\)).

The expansion pass of the mode superposition transient analysis involves computing the displacements at slave DOFs if the reduced modal analysis was used and computing element stresses.

Nodal load output consists of the static loads only as described for a static analysis. The reaction load values represent the negative of the sum of the static loads over all elements connected to a given fixed displacement node. Damping and inertia forces are not included in the reaction loads.
CHAPTER THREE
DYNAMIC ANALYSIS OF BEAMS
UNDER MOVING LOAD

3.1 Introduction

The problem of moving load is considered to be the basis of the dynamic loading and vibration conditions of many different engineering structures. Railway tracks, bridge constructions, rolling bearings, pneumatic and hydraulic cylinders are all some complex examples of engineering applications which involves the moving load problem inside. Researches that make studies on these subjects, usually, begin to their studies, with a beam subjected to a moving load which is known as the basic type of moving load problems. The dynamic behavior of beam structures subjected to a moving load or masses has been investigated for many years. In the early years of moving load studies, the theory and the solution by analytical methods are widely studied. But now, owing to development in the computer and equipment technology, researchers use finite element methods by computer aided engineering (CAE) in their studies. In this study the CAE software ANSYS, is used for the vibration analysis. The vibration analysis of a beam subjected to moving load by the software I-DEAS is studied by Kiral (2002). He investigated the dynamic response of a moving singular force (F) on a simple-supported beam in the transverse direction from one end to another. He also compared the results obtained by I-DEAS with the results of Olsson (1991) who use analytic methods. In this study, these past results are obtained by using ANSYS. In addition to this, a further analysis is done about the effects of the cycle number of the moving load in the dynamic analysis.
3.2 Beam Subjected to Moving Load

A simple-supported uniform beam subjected to the excitation of a moving load is considered. The physical properties of the beam’s material and the geometrical data are taken similar to the data in the study made by Kiral (2002) in order to compare the results. The cross-sectional area \((b \times h)\) and the overall length \((L)\) of the beam are \(2.0 \times 10^{-4} \text{ m}^2\) and \(0.5 \text{ m}\), respectively. The beam has a flexural rigidity of \(EI = (2.06 \times 10^{11}) \times ((0.01 \times 0.02^3)/12) \text{ N-m}^2\) and a material density of \(7800 \text{ kg/m}^3\). The moving singular force \((F)\) has a magnitude of \(-100 \text{ N}\) in the transverse direction. The velocity of the moving load is constant, \(v = L/\tau\), where \(\tau\) is the traveling time across the beam span.

The beam is modeled by using the BEAM3 element type by ANSYS and divided into 20 equal beam elements (Figure 3.1). The beam is simply-supported. The first and the last nodes are constrained as shown in Figure 3.1.

---

Figure 3.1 Finite element model of the beam
3.3 Solution

All analysis is performed in the solution task of ANSYS. Three analyses are applied to the finite element beam model. First of all, static analysis is done and the deflection of the beam is obtained under single force acting on to the mid-node of the beam as seen from Figure 3.2.

![Figure 3.2 Static loading model of the beam](image)

The static deflection of the beam is calculated to normalize the deflection of the beam under the moving load obtained from the dynamic transient analysis. The nodal deflection results of the beam under a single force are shown in Figure 3.3.

![Figure 3.3 Static deflection of the beam](image)
The maximum deflection of the beam is obtained as $0.189 \times 10^{-3}$ m in the -y direction.

The second step is to perform a modal analysis. 10 vibration modes and corresponding mode shapes are calculated for the dynamic response of the beam under a moving load. 10 natural frequencies are given in the following table (Table 3.1).

<table>
<thead>
<tr>
<th></th>
<th>Natural frequencies of the beam</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>186.3 Hz</td>
</tr>
<tr>
<td>2</td>
<td>743.75 Hz</td>
</tr>
<tr>
<td>3</td>
<td>1668 Hz</td>
</tr>
<tr>
<td>4</td>
<td>2570.2 Hz</td>
</tr>
<tr>
<td>5</td>
<td>2952.2 Hz</td>
</tr>
<tr>
<td>6</td>
<td>4587 Hz</td>
</tr>
<tr>
<td>7</td>
<td>6561.3 Hz</td>
</tr>
<tr>
<td>8</td>
<td>7726.5 Hz</td>
</tr>
<tr>
<td>9</td>
<td>8862.5 Hz</td>
</tr>
<tr>
<td>10</td>
<td>11477 Hz</td>
</tr>
</tbody>
</table>

3.3.1 Selection of Time Intervals

While selecting the time intervals used in the dynamic analysis, the highest natural frequency mode is taken into account. By this way the time intervals of the analysis will be small enough to accurately obtain the dynamic response of the beam. This method is especially useful for the systems having small degrees of freedom. For large order systems, it is clear that very small time intervals will increase the time of the analysis and also it will make the analysis difficult to be solved. Using the method of mode-superposition is a way to deal with this problem. The mode-superposition method makes it possible to use only the modes which have priority over other modes in that dynamic analysis. The buckling modes can be more important for some analysis like a dynamic response of a beam subjected to a moving load, while the expansion modes can be more significant for a transient
analysis of a cylinder subjected to a moving internal pressure. Although the mode-
superposition method has many advantages, because of the simplicity of the beam
model, all modes are taken into consideration in this study.
Consequently, the time interval is chosen as \( \Delta t = \frac{T^{10}}{20} \) during the analysis in
order to ensure that all the 10 modes contribute to the dynamic response, where \( T^{10} \)
is the period of the 10\(^{th}\) natural mode of the beam.

3.3.2 Excitation Functions

One of the most important points of a dynamic analysis is to build the excitation
functions for the nodal force moving in the transverse direction. There are some
different methods to form these functions in the solution task of ANSYS. In fact,
using the appropriate command is the only way to create the functions but, ANSYS
allows users to activate the command in some different methods in general. One way
is to use the graphical user interface (GUI) to use the commands and another way is
to create a file of commands which can be read into ANSYS as an input data. By this
way, all the proper commands are executed consecutively as they functioned one
after another in GUI.

Entering the commands from a file also makes it possible to use the process
control (APDL) commands. APDL commands are used to control the order in which
other commands are processed. Cycle, do-endo, if-else if-endif, go and etc. are
some examples of APDL commands. By using these commands, the time histories
for all nodes are defined in a text file and read into ANSYS as the excitation
functions for the dynamic analysis. The time history of the nodal force is given in
Figure 3.4.
\( \tau_i = \frac{v}{x_i} \), where \( x_i \) is the location of the \( i \)\(^{th}\) node.

3.3.3 Transient Analysis

In the past studies, the time histories for the normalized mid-span displacements
are calculated by the analytical method (Olsson, 1991). The same dynamic response
is also obtained by I-DEAS (Kiral, 2002). A single cycle of the moving load from one end to another is investigated in their studies.

The time history results of the single cycle solution obtained by ANSYS are shown in Figure 3.6. In addition to this the dynamic response for double cycle is calculated as well. The gathered results of single and double cycle analysis are compared with each other to examine the effect of the number of cycles of the moving load. The “cycle” definition is given in Figure 3.5.

![Figure 3.4 Traveling load time history for the $i$th node](image)
The deflection \( (u_d) \) under the moving load is normalized with the static deflection \( (u_s) \) when the load is at mid-span. The non-dimensional parameter \( \alpha_p \) equals to \( T_1/2\tau \), where \( T_1 \) is the first natural period. For the cases examined in this study, the velocity of the moving load is assigned to different values by changing the non-dimensional parameter between 0.1 and 1. There are 80 different \( \alpha_p \) values between 0.1 and 1 are used in the analysis. \( D_d \) is called as the dynamic magnification factor and is defined as the ratio of the maximum displacement in the time history of the mid-span to the static mid-span displacement. The results for \( D_d \) of the single cycle analysis obtained by ANSYS (Figure 3.7) are compared with the results obtained by I-DEAS (Kiral, 2002). It is observed from Figures 3.6 and Figure 3.7 that the results obtained by ANSYS are approximately same as the results obtained by I-DEAS.
Figure 3.6 Time history for normalized mid-span displacement obtained by ANSYS, $\alpha_p = 0.25$, single cycle

Figure 3.7 Dynamic magnification factor-versus-$\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1, single cycle
Figure 3.8 shows the ratio of the time ($t_{\text{max}}$) at which the maximum value of the mid-span displacement occurs to the traveling time across the beam for different $\alpha_p$ values.

![Graph showing normalized time for the maximum mid-span displacements versus $\alpha_p$](image)

**Figure 3.8** Normalized time for the maximum mid-span displacements-versus-$\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1, single cycle

The dynamic analyses for one cycle are repeated for two cycles to investigate the effect of number of cycles of the moving load on the beam. Results for $D_d$ of the double cycle analysis obtained by ANSYS are shown in Figure 3.9. It can be seen from Figure 3.9 that in the second cycle of the moving load, the dynamic deflections of the beam are increased. About the half-way of the second cycle, when the moving load is passing over the mid-span, the dynamic deflection is reached to its peak value.
Figure 3.9 Time history for normalized mid-span displacement obtained by ANSYS, $\alpha_p = 0.25$, double cycle

Figure 3.10 shows the values of dynamic magnification factor for different $\alpha_p$ values. In the Figure 3.11, the ratio of the time ($t_{\text{max}}$) to the traveling time across the beam for different $\alpha_p$ values is plotted. $T_{\text{max}}$ is the time at which the maximum value of the mid-span displacement occurs. The cycles corresponding to the time of the force F are also stated in the time scale of the Figure 3.11. It can be observed from the figure that for the $\alpha_p$ values less than 0.5, the maximum mid-span displacements occurs approximately when the force F is around the mid-span in the cycle 2. But there is an exception to this situation at a $\alpha_p$ value of 0.2. As it can be seen from Figure 3.12 that there are four maximum points of mid-span displacements. The first two of them are in the first cycle at the time $0.44 \times 10^{-2}$ s and $0.89 \times 10^{-2}$ s. The other two ones are occurred at the time $0.178 \times 10^{-2}$ s and $0.22 \times 10^{-2}$ s. Although the exact maximum point is occurred at the time $0.44 \times 10^{-2}$ s, the rest of them have approximately the same values of displacements. For this reason, the minima of time ratio in Figure 3.11 can be corrected as Figure 3.13.
Figure 3.10  Dynamic magnification factor versus $\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1 , double cycle

Figure 3.11  Normalized time for the maximum mid-span displacements versus $\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1 , double cycle
Figure 3.12 Time history of normalized mid-span displacement for $\alpha_p = 0.20$

Figure 3.13 Normalized time for the maximum mid-span displacements versus $\alpha_p$ obtained by ANSYS, $\alpha_p$ is on logarithmic scale 0.1-1, double cycle, corrected
It can be seen from Figure 3.13 that for the values of $\alpha_p$ less than 0.6, the maximum mid-span displacement occurs approximately when the force $P$ is around the mid-span in the second cycle. For $\alpha_p$ values greater than 0.6, the maximum mid-span displacement occurs when the force is close to supports at the ends of the first and second cycles.
CHAPTER FOUR  
DYNAMIC ANALYSIS OF HYDRAULIC CYLINDER SUBJECTED TO MOVING PRESSURE  

4.1 Introduction  

Hydraulic and pneumatic system equipments are the important components of engineering applications. Especially hydraulic and pneumatic cylinders are used in many engineering applications like; automatic manufacturing and montage lines, heavy construction equipments, control systems, sensitive measurement and test systems.  

One of the most important factors considering at the design step of these equipments is working conditions of cylinder. Cylinders have different working frequencies according to their usage fields. While the huge sized cylinders used in systems that requires higher force and moment inputs, works generally in lower frequencies, the small sized cylinders used in sensitive application fields like test and measurement systems can have higher working frequencies.  

At the lower working frequency situations, pressure effect on the cylinder is considered as static load, and the hydraulic system equipments are designed according to this criterion. Besides this, at the design procedure of cylinders with higher working frequencies, the dynamic effect with respect to instantaneous change of pressure must be taken into consideration as well as the static analysis.
When the dynamic loading on cylinder is investigated, it is seen that, cylinder surface area subjected to the hydraulic pressure increases with respect to time while the piston travels from one end of the cylinder to the other and it reaches to the highest value at the end of the stroke. Despite the forward movement of the piston, the loaded surfaces are unloaded until the piston comes over that region and these surfaces are reloaded with the passing of the piston.

Loading in the cylinders with the present conditions can be defined as a moving load with some differences.

In this study, hydraulic and pneumatic system components are modeled with the computer aided design programs and dynamic analysis performed by using finite element methods. First of all, a finite element model of a double-acting hydraulic cylinder is created by ANSYS software. Then by performing a static analysis, the static displacements under the working pressure of cylinder are obtained with respect to boundary conditions. Nodal force functions formed by pressure changes on nodes under the working conditions are developed with a written ANSYS software code. In the next step, natural frequencies of the cylinder and mode shapes are obtained. Finally, dynamic loading of the cylinder is formed by considering the force functions and dynamic analysis which is realized according to this loading condition. Dynamic magnification factor of the cylinder which corresponds to the ratio of dynamic results’ displacements to static results’ displacements is investigated according to the working frequency of cylinder and boundary conditions.

And also the effects of damping on the system are investigated parametrically during the dynamic analysis.

4.2 Modeling of the Hydraulic Cylinder

APDL language of the ANSYS software is used for creating the model and performing the analysis of this study. APDL stands for ANSYS Parametric Design Language, a scripting language that one can use to automate the common tasks or
even building the model in term of parameters. APDL also includes a wide range of other features such as repeating a command, macros, if-then-else, do-loop cycles, and scalar, vector or matrix operations.

Every important modeling and analysis parameter used in these codes is parameterized. This makes it possible to analyze many different situations. APDL codes used in this study are summarized in the Appendix 2.

In this thesis, a double-acting hydraulic cylinder is used in the analysis. The model shown in Figure 4.1 is created as finite elements to ensure the variation of parameters easily when is needed instead of using any solid model software. Nodes are created by do-loop and if-then cycles and the elements are created from nodes in the APDL codes in the preparation of the model.

Hydraulic cylinder, modeled as finite element, is made of steel. The properties of steel is given at Table 4.1.

<table>
<thead>
<tr>
<th>Table 4.1 Material properties of cylinder</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of Elasticity, E</td>
</tr>
<tr>
<td>Poisson Ratio, ν</td>
</tr>
<tr>
<td>Density, ρ</td>
</tr>
</tbody>
</table>

In this finite element model, Solid45 element with eight nodes and three degrees of freedom in each node is chosen from ANSYS element library.

All the dimensions of the cylinder are parametric and the dimensions used in the analysis are shown in Figure 4.1 as follows:
Figure 4.1 The finite element model and the cylinder dimensions

In the analysis, two different boundary conditions are considered for the hydraulic cylinder. It is assumed that, the cylinder is directly installed to the other components or to the body in two different mounting types and the static and dynamic analyses is made for these two mounting types. The degree of freedom boundary condition is shown in Figure 4.2.

Figure 4.2 Boundary condition a)Bc-type 1
Working pressure of cylinder is taken as 250 bars and as it is seen in Figure 4.3, the pressure load is applied statically to the inner surface of cylinder. The region that pressure applied is considered as stroke region of the piston.

Figure 4.2 Boundary condition b) Bc-type 2

Figure 4.3 Cylinder filled with 250 bar pressure a) isometric view
4.3 Static Analysis

For the hydraulic cylinder, the static loading condition is defined with applying the pressure to the inner surface of the cylinder. The pressure loading to the inner surface is considered by the arrival of the piston to the end of the stroke and the filling the cylinder with the fluid. Static analyses are made with APDL codes and static displacement values of the nodes are obtained for both of the boundary conditions.

The dynamic magnification factor value is obtained by taking the ratio of the static displacement values to the dynamic displacement values for the same points derived by vibration analysis results. Dynamic magnification factor is an important design parameter which shows the relationship between loading velocity in the moving load problem and the dynamic response of the construction. Dynamic
magnification factor is defined as the ratio of the dynamic displacement values to the static displacement values for a specific point.

These analyses are made in the static analysis module under ANSYS solution task. The stiffness matrix of the system is obtained with the static analysis by using the finite element method. Displacement values for every node are calculated by using the equation below under specific loading. The overall equilibrium equations for linear structural static analysis are:

\[ [K][u] = [F] \]  

where:

- \([K]\) = Total stiffness matrix
- \([u]\) = Nodal displacement vector
- \([F]\) = Total applied load vector

The effects of the steady loading conditions and the displacement values of the whole structure are determined by the static analysis.

Figure 4.4 and Figure 4.5 show the displacement distribution in the cylinder under the 250 bar pressure for both boundary conditions. The node 1522 and the node 2809 are selected for the analysis. The node 1522 is at the half-way of the stroke of the pressure loading and has 45° angular position. The node 2809 is at the beginning of the stroke and 0° angular position. Static displacement values for the investigated nodes for the two boundary conditions are shown in Table 4.2.

<table>
<thead>
<tr>
<th>Table 4.2 Static displacement values of nodes 1522 and 2809</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Node 1522, 45°</strong></td>
</tr>
<tr>
<td>----------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Bc-type1</strong></td>
</tr>
<tr>
<td>(u_x ) (m)</td>
</tr>
<tr>
<td>0.152E-03</td>
</tr>
<tr>
<td>(u_y ) (m)</td>
</tr>
<tr>
<td>0.152E-03</td>
</tr>
<tr>
<td>(U_{\text{total}} ) (m)</td>
</tr>
<tr>
<td>0.215E-03</td>
</tr>
</tbody>
</table>
Figure 4.4 Static displacement distribution of cylinder, Bc-type1

Figure 4.5 Static displacement distribution of cylinder, Bc-type2
4.4 Vibration Analysis

Vibration analysis is investigated in three subtitles.

- Formation of loading functions.
- Modal analysis
- Transient analysis

4.4.1 Formation of Loading Functions

In the static analysis, the static displacements are calculated to compute the dynamic magnification factor. Loading condition will be different from static state when hydraulic cylinder is considered as double-acting and periodic working conditions. To perform the dynamic analysis, a loading model is required to ensure periodic study. This loading function is formed with APDL language and the loading functions obtained are used both in the “static.txt” APDL code and “transient.txt” APDL code. (Appendix 2)

4.4.2 Modal Analysis

Natural frequencies of the system and mode shapes have to be calculated for the full solution method and mode superposition method which will be used in dynamic analysis step. Response of the system for the external forcing is determined by forcing frequency and vibration shapes. According to these methods, initially the model analysis is done for the present boundary conditions of the cylinder and 20 natural frequencies and vibration shapes are obtained for both boundary conditions. Natural frequencies for both boundary conditions are shown in Table 4.3 and Table 4.4. Two vibration shape examples for both boundary condition of investigated hydraulic cylinder are shown in Figure 4.6, Figure 4.7, Figure 4.8, and Figure 4.9.
Table 4.3 Natural Frequencies of Cylinder, Bc-type1

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Natural Frequency(Hz)</th>
<th>Mode Number</th>
<th>Natural Frequency(Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>137.30</td>
<td>11.</td>
<td>1138.0</td>
</tr>
<tr>
<td>2.</td>
<td>137.30</td>
<td>12.</td>
<td>1138.0</td>
</tr>
<tr>
<td>3.</td>
<td>576.85</td>
<td>13.</td>
<td>1327.8</td>
</tr>
<tr>
<td>4.</td>
<td>576.85</td>
<td>14.</td>
<td>1327.8</td>
</tr>
<tr>
<td>5.</td>
<td>595.82</td>
<td>15.</td>
<td>1429.6</td>
</tr>
<tr>
<td>6.</td>
<td>852.09</td>
<td>16.</td>
<td>1429.6</td>
</tr>
<tr>
<td>7.</td>
<td>852.09</td>
<td>17.</td>
<td>1708.9</td>
</tr>
<tr>
<td>8.</td>
<td>1023.9</td>
<td>18.</td>
<td>1708.9</td>
</tr>
<tr>
<td>9.</td>
<td>1060.2</td>
<td>19.</td>
<td>1938.3</td>
</tr>
<tr>
<td>10.</td>
<td>1060.2</td>
<td>20.</td>
<td>1938.3</td>
</tr>
</tbody>
</table>

Table 4.4 Natural Frequencies of Cylinder, Bc-type2

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Natural Frequency(Hz)</th>
<th>Mode Number</th>
<th>Natural Frequency(Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>623.71</td>
<td>11.</td>
<td>1724.5</td>
</tr>
<tr>
<td>2.</td>
<td>623.71</td>
<td>12.</td>
<td>1724.5</td>
</tr>
<tr>
<td>3.</td>
<td>1074.6</td>
<td>13.</td>
<td>1902.7</td>
</tr>
<tr>
<td>4.</td>
<td>1074.6</td>
<td>14.</td>
<td>1998.9</td>
</tr>
<tr>
<td>5.</td>
<td>1139.2</td>
<td>15.</td>
<td>1998.9</td>
</tr>
<tr>
<td>6.</td>
<td>1139.2</td>
<td>16.</td>
<td>2128.2</td>
</tr>
<tr>
<td>7.</td>
<td>1226.2</td>
<td>17.</td>
<td>2128.2</td>
</tr>
<tr>
<td>8.</td>
<td>1226.2</td>
<td>18.</td>
<td>2222.1</td>
</tr>
<tr>
<td>9.</td>
<td>1334.1</td>
<td>19.</td>
<td>2222.1</td>
</tr>
<tr>
<td>10.</td>
<td>1334.1</td>
<td>20.</td>
<td>2225.0</td>
</tr>
</tbody>
</table>
Figure 4.6 Mode Shape Example, Bc-type1
Mode 3 (576.85Hz)

Figure 4.7 Mode Shape Example, Bc-type1
Mode 9 (1060.2Hz)
Figure 4.8 Mode Shape Example, Bc-type2
Mode 1 (623.71Hz)

Figure 4.9 Mode Shape Example, Bc-type2
Mode 5 (1139.2Hz)
4.4.3 Transient Analysis

The transient analysis of the hydraulic cylinder is performed in the solution task of the CAE program ANSYS. The set of equations for the forced and undamped vibration of a system by using the finite element methods is formed as follows;

\[
[M][\ddot{u}] + [K][u] = \{f(t)\}
\]

(4.2)

where; \([M]\) is the mass matrix,
\([\ddot{u}]\) is the acceleration vector,
\([u]\) is the displacement vector
and \([f(t)]\) is the time-dependent force vector defined for the nodes subjected to pressure loading.

Both the full solution method and the mode-superposition method are used in the transient analysis. Although, set of desired mode-shapes can be considered in the mode-superposition method, all extracted mode shapes are taken into consideration in the solution of the system. By considering all extracted modes in the mode-superposition method, a similar solution to the full solution is done and also more shorter run-times are obtained with respect to full solution method.

The vibration analysis is studied between the working frequencies 150Hz and 1150 Hz. And the time interval is selected as:

\[
\Delta t = T_{12} / 20
\]

where \(T_{12}\) is the period of the mode shape 12.

A value of 0.002 is used for \(a\) and \(\beta\) damping coefficient in the dynamic analysis.
As the cycle definition for the beam is given in chapter 3, a similar approach is followed in the transient analysis of the hydraulic cylinder. All transient analysis in this chapter is done for four consecutive cycles. Eleven separate solutions are done and eleven different deflection values under the moving pressure are calculated for the nodes under investigation. The deflection ($u_i$) under the moving pressure of nodes 1522 and 2809 are normalized with the static deflections ($u_s$) of nodes 1522 and 2809, respectively. The dynamic magnification factors ($D_d$) are obtained as a result of this normalization. Figure 4.10 and 4.11 shows the time history for normalized displacement values of node 1522 for both of the two boundary conditions. And Figure 4.12 and 4.13 shows the time history for normalized displacement values of node 2809 for both of the two boundary conditions. As it can be easily seen from Figures 4.10 to 4.13 the deflection values of the hydraulic cylinder under the moving internal pressure are at least one and a half times larger than the static deflection values.
Figure 4.11 Time history for normalized displacements of node 1522, Bc-type2

Figure 4.12 Time history for normalized displacements of node 2809, Bc-type1
Figure 4.13 Time history for normalized displacements of node 2809, Bc-type2

Figure 4.14 Normalized time for the maximum mid-stroke displacements-versus-frequency, Bc-type1
Figure 4.14 shows the ratio of the time ($T_{\text{max}}$) at which the maximum value of the mid-stroke displacement occurs to the traveling time across the cylinder for different frequency values. In this figure, every quarter of the $T_{\text{max}} / T_{\text{total}}$ scale, corresponds one cycle of moving internal pressure loading. It can be seen from Figure 4.14 that for the frequencies up to 350 Hz, the maximum deflection occurs in third and the fourth cycles, the maximum deflection occurs in the first two cycles for the frequencies bigger than 450 Hz.

4.4.4 The effect of damping in the transient dynamic analysis

One of the most important parameters used in the transient dynamic analysis is damping parameter. To investigate the effect of damping in the dynamic analysis, eight different damping coefficient values are used at a working frequency of 550 Hz and eight different maximum displacement values are calculated for node 1522. It can be observed from Figure 4.15 that maximum displacement values increase as the damping values becomes smaller. Also for the damping coefficient values such as 0.0001 and 0.0005, a small amount of difference can be observed. So for this analysis, it can be suggested that the damping values have no effect for the values smaller than 0.0001.

<table>
<thead>
<tr>
<th>Damping Coefficients</th>
<th>Max. Displacements (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.1765E-3</td>
</tr>
<tr>
<td>0.1</td>
<td>0.2304E-3</td>
</tr>
<tr>
<td>0.05</td>
<td>0.3042E-3</td>
</tr>
<tr>
<td>0.01</td>
<td>0.4588E-3</td>
</tr>
<tr>
<td>0.005</td>
<td>0.4934E-3</td>
</tr>
<tr>
<td>0.001</td>
<td>0.5243E-3</td>
</tr>
<tr>
<td>0.0005</td>
<td>0.5283E-3</td>
</tr>
<tr>
<td>0.0001</td>
<td>0.5302E-3</td>
</tr>
</tbody>
</table>
Vibration analysis of the hydraulic system components for the different speed, size and the loading conditions can be performed by using the commercial finite element packages if the dynamic loading of the structure is properly defined. The APDL commands of the computer aided engineering program ANSYS is used to write codes for defining both the structural parameters and the dynamic loading of the beam and the hydraulic cylinder. Constructing the written code in a manner to allow any parametrical changes of the structural dimensions or the loading condition provides a significant amount of saving in time, work power and money. By this way, many different cases can be successfully applied to the analysis of the structure under investigation.

From the results presented in this study, the following points of discussions are summarized:

- The commercial finite element packages can be used to analyze the vibration of the engineering structures subjected to moving loads with the proper definition of the dynamic loading. A simply supported beam subjected to singular moving load is analyzed with the finite element package ANSYS. Good agreement is found between the results obtained by ANSYS and the results obtained from another FE package I-DEAS presented in the literature.
• The dynamic magnification factor and the critical moving load speeds can be determined during the design stage with high precision of engineering structure.

• The dynamic magnification factor increases as the number of the loading cycles of the singular moving load on the simply supported beam increases. The maximum deflection of the mid-span of the beam under dynamic loading occurs mostly in the last cycle of loading when the moving load passes over the mid-span of the beam.

• It is possible to mention about the dynamic magnification factor for three-dimensional structures such as hydraulic cylinders.

• The dynamic magnification values are obtained mostly bigger than 2 for the proper band of pressure loading speeds related with the frequencies of the mode shapes under investigation.

• It is observed that the dynamic magnification factor varies with parameters such as boundary conditions, working frequency, system damping.
REFERENCES


APPENDIX 1

The written ANSYS APDL code for the dynamic analysis of the beam:

BEAM-MODEL.TXT

/config,nres,100000
/prep7
l=0.5
b=0.01
h=0.02
damp=0.0003
f0=-100
f1=186.30
f2=743.75
f10=11477
f20=33033
dx=l/20
e=206e9
mp,ex,1,e
mp,dens,1,7800

et,1,beam3
a=b*h
iz=b*h*h*h/12
r,1,a,iz,h/2

k=0
*do,x,0,l,dx
k=k+1
n,k,x
*enddo
nd=k

*do,k,2,nd
e,k-1,k
*enddo
/triad,lbot
!/window,1,top
/plopts,info,1
/plopts,logo,1
/pbc,rfor,0
gplot
d,1,ux,0
d,1,uy,0
d,nd,uy,0

BEAM-STATIC.TXT
nr=(nd+1)/2
f,nr,fy,f0
/solu
antype,static,new
solve
finish

/post1
plnsol,u,y

BEAM-TRANSIENT.TXT
/solu
!alphad,damp
!betad,damp

nr=(nd+1)/2
ncycle=1
tsay=ncycle*(nd-1)
alpha=0.25

t1=1/f1
t0=t1/(2*alpha)
\[
\text{dt} = \frac{t_0}{(nd-1)}
\]
\[
\text{t10} = \frac{1}{f_{10}}
\]
\[
\text{ddt} = \frac{\text{t10}}{(nd-1)}
\]

antype, trans, new
outres, all, all
kbc, 0
deltim, ddt

t=0
x=1
*do, ta, 1, tsay+2-nd, (nd-1)
  *if, ta, eq, x, then
  *do, k, 2, nd
    f,k-1, fy, 0
    f,k, fy, f0
    t=t+dt
    time, t
    solve
    gplot
  *enddo
  x=x+((nd-1)*2)
  *else
  *do, k, nd-1, 1, (-1)
    f,k+1, fy, 0
    f,k, fy, f0
    t=t+dt
    time, t
    solve
    gplot
  *enddo
*endif

gplot
*enddo

finish
/post26
file., rdsp
nsol, 2, nr, u, y
EXTREM, 2
*get, maxy, VARI, 2, extrem, vmin
*get, tmaxy, VARI, 2, extrem, tmin
APPENDIX 2

The written ANSYS APDL code for the dynamic analysis of the hydraulic cylinder:

**CYLD-MODEL.TXT**

/config,nres,100000
/prep7
/view,1,1,1,0
/color,u,blue
/pi=3.141592654
/*afun,deg

!!!INPUTS

r1=0.09
r2=0.095
deltet=5
n1=1
es=20
h=0.8
delh=h/es
deln=(r2-r1)/n1
cs=es-1

et,1,solid45
mp,ex,1,203e9
mp,dens,1,7850
mp,prxy,1,0.3

!MODELING

k=0
*do,hd,0,h,delh
*do,ns,1,n1
  *do,tet,0,(360-deltet),deltet
    k=k+1
    n,k,(r1+(deln*(ns-1)))*cos(tet),{(r1+(deln*(ns-1)))*sin(tet),hd
  *endo
*endo
*do,tet,0,(360-deltet),deltet
k=k+1
n,k,r2*cos(tet),r2*sin(tet),hd
*enddo
*enddo
nd=k/((es+1)*(n1+1))
nz=(n1+1)*nd

*do,ck,0,(es-1)
 *do,nk,0,(n1-1)
  *do,m,2,nd
   m=m+(nk*nd)+(nz*ck)
   e,m,m-1,m+nd-1,m+nd,m+nz,m+nz-1,m+nd+nz-1,m+nd+nz
   *enddo
   e,m-nd+1,m,m+nd,m+1,m+nz-nd+1,m+nz,m+nd+nz,m+nz+1
  *enddo
 *enddo

m=k
kt1=1.2
*do,md,h,h+(h/9),h/9
  *do,tet,0,(360-deltet),deltet
   m=m+1
   n,m,r1*cos(tet),r1*sin(tet),md
  *enddo
  *do,tet,0,(360-deltet),deltet
   m=m+1
   n,m,r2*cos(tet),r2*sin(tet),md
  *enddo

  *do,tet,0,(360-deltet),deltet
   m=m+1
   n,m,kt1*r2*cos(tet),kt1*r2*sin(tet),md
  *enddo
*enddo

ndele,k+1,k+nz
mm=k

*do,m1,2,nd
m1=m1+mm
e,m1-nz,m1-nz-1,m1-nd-1,m1-nd,m1+nz+nd,m1+nz+nd-1,m1+nz+nz-1,m1+nz+nz
e,m1-nd,m1-nd-1,m1+nd,m1+nd-1,m1+nd+nd-1,m1+nd+nd
*enddo

  e,m1-nz+1,m1-nz+1,m1-nd+1,m1-nd,m1+nz+nd+1,m1+nz+nd+1,m1+nz+nd+1,m1+nz+nd+1
pp=m
kt1=1.2
*do,pd,0,(-1)*((h/9),(-1)*((h/9)
  *do,tet,0,(360-deltet),deltet
  pp=pp+1
  n,pp,r1*cos(tet),r1*sin(tet),pd
  *endo
*do,tet,0,(360-deltet),deltet
  pp=pp+1
  n,pp,r2*cos(tet),r2*sin(tet),pd
  *endo
*do,tet,0,(360-deltet),deltet
  pp=pp+1
  n,pp,kt1*r2*cos(tet),kt1*r2*sin(tet),pd
  *endo
*enddo
ndele,m+1,m+nz
ppp=m
*do,m2,2,nd
  m2=m2+ppp
e,m2-ppp,m2-ppp-1,m2-ppp+nd-1,m2-ppp+nd,m2+nz+nd,m2+nz+nd-1,m2+nz+nz-1,m2+nz+nz
e,m2-ppp+nd,m2-ppp+nd-1,m2+nz-1,m2+nz,m2+nz+nz,m2+nz+nz-1,m2+nz+nz+nd
  *endo
e,m2-ppp+1,m2-ppp+1,m2+nd+1,m2+nd+1,m2+nd,m2+nd,m2+nd,m2+nd
  e,m2-ppp+1,m2-
  ppp+nd,m2+nd+1,m2+nd+1,m2+nd+1,m2+nd+1,m2+nd+1,m2+nd+1,m2+nd+1
  gplot

!BOUNDARY CONDITIONS
*do,ds,m+nz+nd+1,m+nz+nz+nz
d,ds,all,0
*endo
!*do,ds,k+nz+nd+1,k+nz+nz+nz
!d,ds,all,0
!*endo
**CYLD-MODAL.TXT**

*afun,deg
/solu
antype,2
modopt,lanb,20
solve
*get,f1,mode,1,freq
*get,f2,mode,2,freq
*get,f3,mode,3,freq
*get,f4,mode,4,freq
*get,f5,mode,5,freq
*get,f6,mode,6,freq
*get,f7,mode,7,freq
*get,f8,mode,8,freq
*get,f9,mode,9,freq
*get,f10,mode,10,freq
*get,f11,mode,11,freq
*get,f12,mode,12,freq
*get,f13,mode,13,freq
*get,f14,mode,14,freq
*get,f15,mode,15,freq
*get,f16,mode,16,freq
*get,f17,mode,17,freq
*get,f18,mode,18,freq
*get,f19,mode,19,freq
*get,f20,mode,20,freq
FINISH

**CYLD-STATIC.TXT**

*afun,deg
P=25000000
A=2*pi*r1*h
tns=nd*cs
f0=P*A/tns
*do,ck,cs,1,(-1)
tet=0
*do,z,1,nd
tet=(z-1)*deltet
z=z+(ck*nz)
f,z,fx,f0*cos(tet)
f, z, fy, \( f_0 \times \sin(\theta) \)
*enddo

*enddo

/solu
antype, static, new
solve
finish
/plopts, leg1, 0
/plopts, logo, 1
/post1
plnsol, u, sum

**CYLD-TRANSIENT.TXT**

*afun, deg
P=25000000
A=2*\( \pi \times r_1 \times h \)
tns=nd*cs
f0=P*A/tns
!damp=0.001
t10=1/f10
ddt=t10/20

ncycle=2
tsay=cs*ncycle

!alpha=0.1
!t5=1/f5
!t0=t5/(2*alpha)
!dt=t0/cs

freqq=5
dt=1/freqq/cs

!TRANSIENT ANALYSIS

/solu
antype, 4
trmnpt, msup, 10,, 1
dmprat,0.005,  
outres,all,all  
kbc,0  
deltim,ddt  
solve  

!alphad=damp  
!betad=damp  
!antype,trans,new  
!outres,all,all  
!kbc,0  
!deltim,ddt  

!LOADING  

t=0  
x=1  
*do,ta,1,tsay-cs+1,cs  
*if,ta,eq,x,then  
  *do,ck,1,cs,1  
  *do,z,1,nd  
    tet=(z-1)*deltet  
    z=z+(ck*nz)  
    f,z,fx,f0*cos(tet)  
    f,z,fy,f0*sin(tet)  
  *enddo  
  t=t+dt  
  time,t  
  solve  
*enddo  
  x=x+(cs*2)  
*else  
  *do,ck,cs,1,(-1)  
  *do,z,1,nd  
    tet=(z-1)*deltet  
    z=z+(ck*nz)  
    f,z,fx,f0*cos(tet)  
    f,z,fy,f0*sin(tet)  
  *enddo  
  t=t+dt  
  time,t  
  solve  
*endif  
/pbc,all,,0
/pbc,u,,1
/rep
nsel,all
fdele,all
allsel,all
!t=t+dt
!time,t
!solve
/pbc,all,,0
/pbc,u,,1
/rep
*enddo
finish
/post26
file,,rdsp
numvar,50
midnxy=(k/2)+(nd/8)+1
endnx=k-nd-nz+1
nsol,2,midnxy,u,x,xmid
nsol,3,midnxy,u,y,ymid
prod,4,2,2
prod,5,3,3
add,6,4,5
sqrt,7,6,,,xymid
nsol,8,endnx,u,x,xend
nsol,9,endnx,u,y,yend
prod,10,8,8
prod,11,9,9
add,12,10,11
sqrt,13,12,,,xyend
plvar,2,7,8