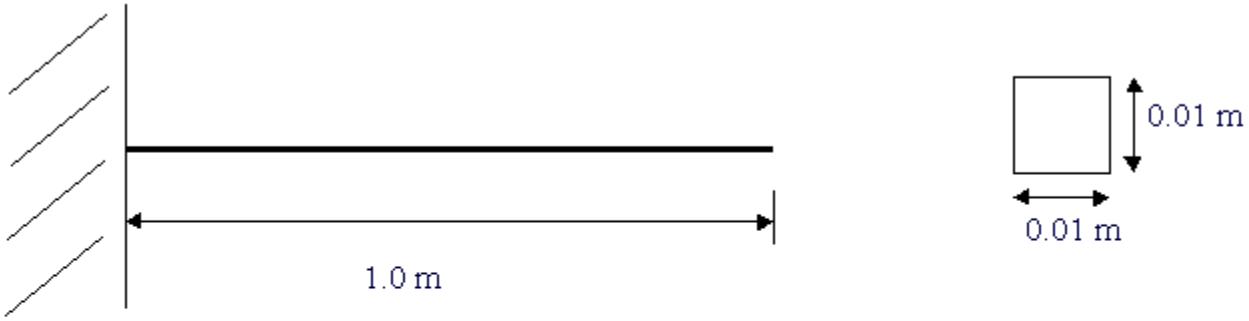
Harmonic Analysis of a Cantilever Beam

Introduction

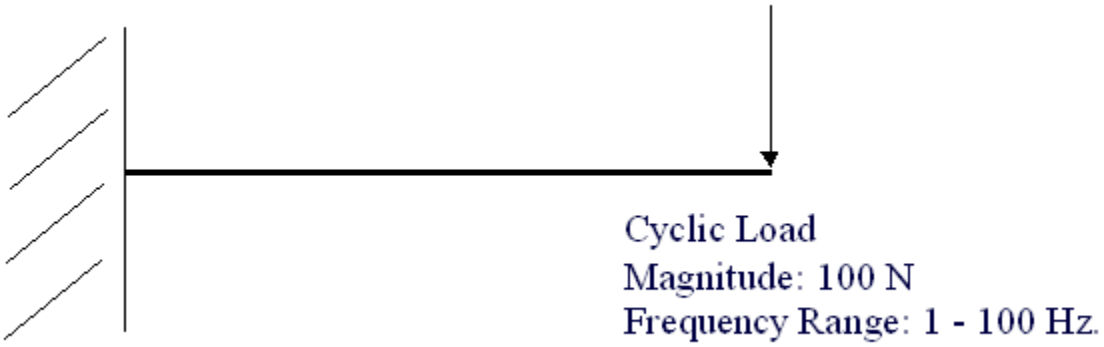
This tutorial was created using ANSYS 7.0 The purpose of this tutorial is to explain the steps required to perform Harmonic analysis the cantilever beam shown below.



Modulus of Elasticity (E) = $206800(10^6)$ N/m^2

Density = 7830 kg/m^3

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 figure below depicts the beam with the application of the load.



ANSYS provides 3 methods for conducting a harmonic analysis. These 3 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.

ANSYS Command Listing

```
FINISH
/CLEAR

/TITLE, Dynamic Analysis
/PREP7

K,1,0,0                          ! Enter keypoints
K,2,1,0

L,1,2                            ! Create line

ET,1,BEAM3                       ! Element type

R,1,0.0001,8.33e-10,0.01         ! Real Const: area,I,height

MP,EX,1,2.068e11                 ! Young's modulus
MP,PRXY,1,0.33                   ! Poisson's ratio
MP,DENS,1,7830                   ! Density

LESIZE,ALL,,,10                  ! Element size
LMESH,1                          ! Mesh line

FINISH
/SOLU

ANTYPE,3                         ! Harmonic analysis

DK,1,ALL                         ! Constrain keypoint 1
FK,2,FY,100                      ! Apply force

HARFRQ,0,100,                    ! Frequency range
NSUBST,100,                      ! Number of frequency steps
KBC,1                            ! Stepped loads

SOLVE
FINISH

/POST26

NSOL,2,2,U,Y, UY_2               ! Get y-deflection data
STORE,MERGE

PRVAR,2                          ! Print data
PLVAR,2                          ! Plot data
```