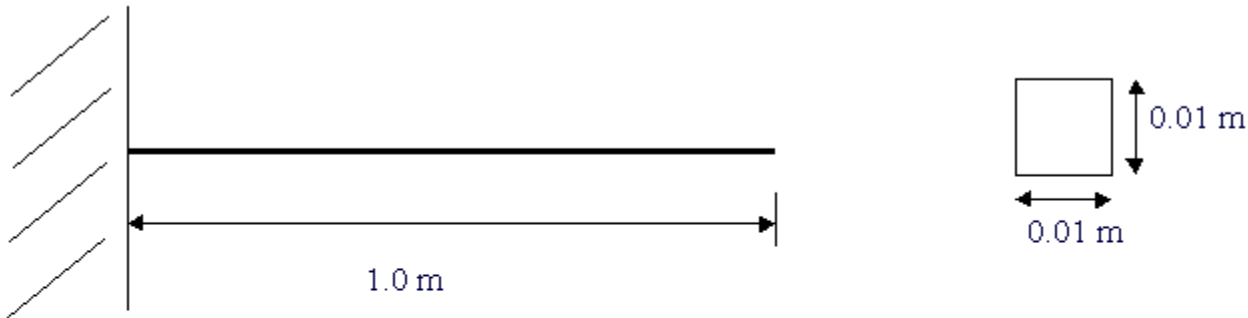


Modal Analysis of a Cantilever Beam

Introduction

This tutorial was created using ANSYS 7.0 The purpose of this tutorial is to outline the steps required to do a simple modal analysis of the cantilever beam shown below.



Modulus of Elasticity (E) = $206800(10^6)$ N/m²

Density = 7830 kg/m³

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,2                ! Modal analysis
MODOPT,SUBSP,5          ! Subspace, 5 modes
EQSLV,FRONT             ! Frontal solver
MXPAND,5                ! Expand 5 modes

DK,1,ALL                ! Constrain keypoint one

SOLVE
FINISH

/POST1                  ! List solutions
SET,LIST

SET,FIRST
PLDISP                  ! Display first mode shape

ANMODE,10,0.5, ,0      ! Animate mode shape
```