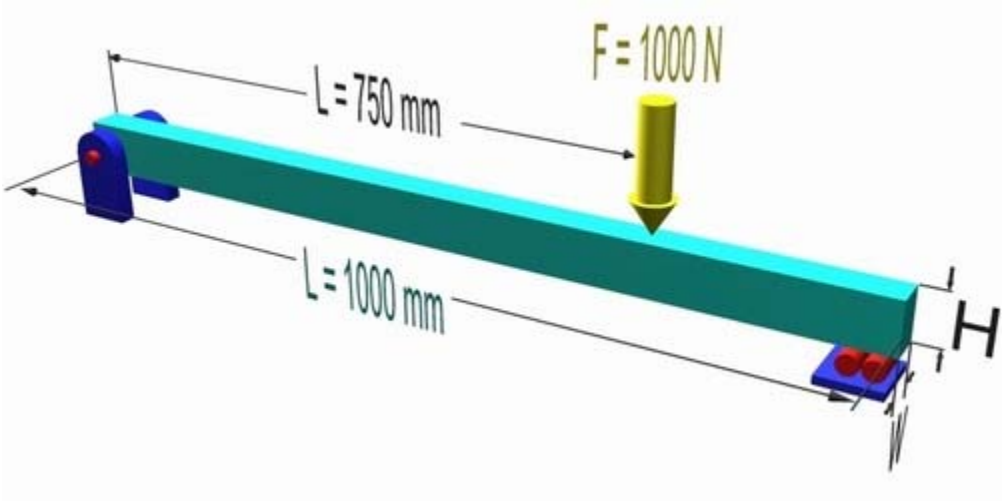
Design Optimization

Introduction

This tutorial was completed using ANSYS 7.0 The purpose of this tutorial is to introduce a method of solving design optimization problems using ANSYS. This will involve creating the geometry utilizing parameters for all the variables, deciding which variables to use as design, state and objective variables and setting the correct tolerances for the problem to obtain an accurately converged solution in a minimal amount of time. The use of hardpoints to apply forces/constraints in the middle of lines will also be covered in this tutorial.

A beam has a force of 1000N applied as shown below. The purpose of this optimization problem is to minimize the weight of the beam without exceeding the allowable stress. It is necessary to find the cross sectional dimensions of the beam in order to minimize the weight of the beam. However, the width and height of the beam cannot be smaller than 10mm. The maximum stress anywhere in the beam cannot exceed 200 MPa. The beam is to be made of steel with a modulus of elasticity of 200 GPa.



ANSYS Command Listing

```
/prep7
/title, Design Optimization
*set,H,20                              ! Set an initial height of 20 mm
*set,W,20                              ! Set an initial width of 20 mm

K,1,0,0                                ! Keypoint locations
K,2,1000,0
L,1,2                                  ! Create line
HPTCREATE,LINE,1,0,RATI,.75,           ! Create hardpoint 75% from left side

ET,1,BEAM3                             ! Element type
R,1,W*H,(W*H**3)/12,H,,,,              ! Real consts: area,I (note '**', not '^'),height
MP,EX,1,200000                         ! Young's modulus
MP,PRXY,1,0.3                          ! Poisson's ratio
```

```
ESIZE,100                               ! Mesh size
LMESH,ALL                               ! Mesh line

FINISH
/SOLU

ANTYPE,0                                ! Static analysis

DK,1,UX,0                               ! Pin keypoint 1
DK,1,UY,0
DK,2,UY,0                               ! Support keypoint 2

FK,3,FY,-2000                           ! Force at hardpoint

SOLVE
FINISH

/POST1
ETABLE,EVolume,VOLU,                    ! Volume of single element
SSUM                                    ! Sum all volumes
*GET,Volume,SSUM,,ITEM,EVOLUME          ! Create parameter 'Volume' for volume of beam

ETABLE,SMAX_I,NMISC,1                   ! Create parameter 'SMaxI' for max stress at I nod
ESORT,ETAB,SMAX_I,0,1,,
*GET,SMAXI,SORT,,MAX

ETABLE,SMAX_J,NMISC,3                   ! Create parameter 'SMaxJ' for max stress at J nod
ESORT,ETAB,SMAX_J,0,1,,
*GET,SMAXJ,SORT,,MAX

*SET,SMAX,SMAXI>SMAXJ                   ! Create parameter 'SMax' as max stress

LGWRITE,optimize,txt,C:\TEMP            ! Save logfile to C:\Temp\optimize.txt

/OPT
OPANL,'optimize','txt','C:\Temp\'       ! Assign optimize.txt as analysis file

OPVAR,H,DV,10,50,0.001                  ! Height design variable, min 10 mm, max 50 mm, to
OPVAR,W,DV,10,50,0.001                  ! Width design variable, min 10 mm, max 50 mm, tol
OPVAR,SMAX,SV,195,200,0.001             ! Height state variable, min 195 MPa, max 200 MPa,
OPVAR,VOLUME,OBJ,,,200                  ! Volume as object variable, tolerance 200 mm^2

OPTYPE,FIRS                             ! First-order analysis
OPFRST,30,100,0.2,                      ! Max iteration, Percent step size, Percent forwar
OPEXE                                   ! Run optimization

PLVAROPT,H,W                            ! Graph optimation data
/AXLAB,X,Number of Iterations
/AXLAB,Y,Width and Height (mm)
/REPLOT
```