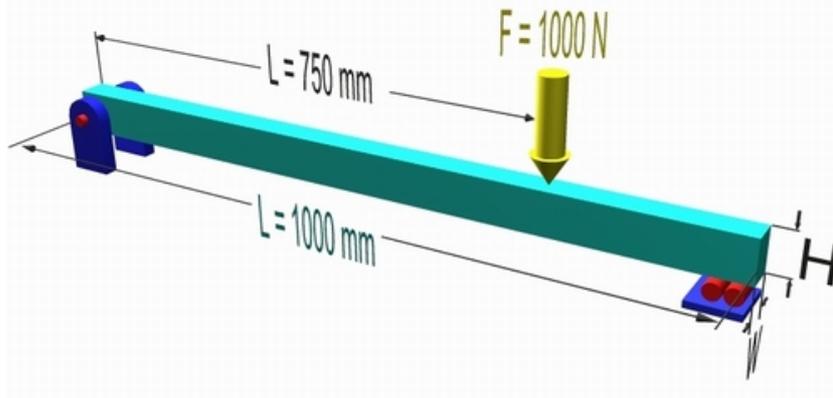# Design Optimization

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.

## Preprocessing: Defining the Problem

1. **Give example a Title**

    Utility Menu > File > Change Title ...
    ```
    /title, Design Optimization
    ```

2. **Enter initial estimates for variables**

    To solve an optimization problem in ANSYS, parameters need to be defined for all design variables.

    o Select: **Utility Menu > Parameters > Scalar Parameters...**
    o In the window that appears (shown below), type `W=20` in the 'Selection' section
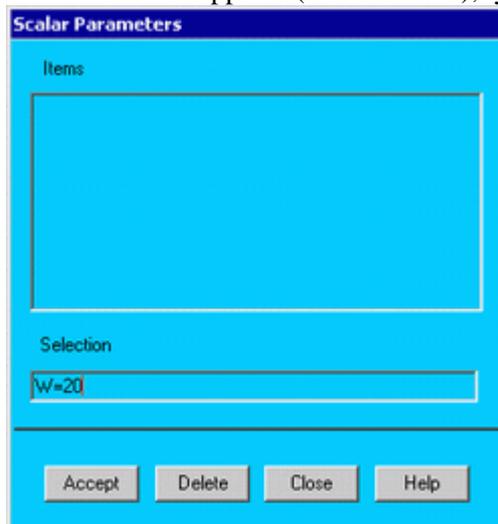
    o Click 'Accept'. The 'Scalar Parameters' window will stay open.
    o Now type `H=20` in the 'Selection' section

١

o Click 'Accept'
o Click 'Close' in the 'Scalar Parameters' window.
   NOTE: None of the variables defined in ANSYS are allowed to have negative values.

3. **Define Keypoints**

   Preprocessor > Modeling > Create > Keypoints > In Active CS...
   K,#,x,y

   We are going to define 2 Keypoints for this beam as given in the following table:

Keypo ints	Coordinates (x,y)
1	(0,0)
2	(1000,0)

4. **Create Lines**

   Preprocessor > Modeling > Create > Lines > Lines > In Active Coord
   L,1,2

   Create a line joining Keypoints 1 and 2

5. **Create Hard Keypoints**

   Hardpoints are often used when you need to apply a constraint or load at a location where a keypoint does not exist. For this case, we want to apply a force 3/4 of the way down the beam. Since there are not any keypoints here and we can't be certain that one of the nodes will be here we will need to specify a hardpoint

   o Select **Preprocessor > Modeling > Create > Keypoints > Hard PT on line > Hard PT by ratio**. This will allow us to create a hardpoint on the line by defining the ratio of the location of the point to the size of the line
   o Select the line when prompted
   o Enter a ratio of 0.75 in the 'Create HardPT by Ratio window which appears.
   You have now created a keypoint labelled 'Keypoint 3' 3/4 of the way down the beam.

6. **Define Element Types**

   Preprocessor > Element Type > Add/Edit/Delete...

   For this problem we will use the BEAM3 (Beam 2D elastic) element. This element has 3 degrees of freedom (translation along the X and Y axes, and rotation about the Z axis).

7. **Define Real Constants**

   Preprocessor > Real Constants... > Add...

   In the 'Real Constants for BEAM3' window, enter the following geometric properties: (Note that '**' is used instead '^' for exponents)
   i. Cross-sectional area AREA: W*H
   ii. Area moment of inertia IZZ: (W*H**3)/12
   iii. Thickness along Y axis: H

   NOTE: It is important to use independent variables to define dependent variables such as the moment of inertia. During the optimization, the width and height will change for each iteration. As a result, the other variables must be defined in relation to the width and height.

8. **Define Element Material Properties**

   Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic
   In the window that appears, enter the following geometric properties for steel:
   . Young's modulus EX: 200000
   i. Poisson's Ratio PRXY: 0.3

9. **Define Mesh Size**

   Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines...
   For this example we will specify an element edge length of 100 mm (10 element divisions along the line).

10. **Mesh the frame**

    Preprocessor > Meshing > Mesh > Lines > click 'Pick All'
    LMESH,ALL

Solution Phase: Assigning Loads and Solving

1. **Define Analysis Type**

   Solution > Analysis Type > New Analysis > Static

   `ANTYPE,0`

2. **Apply Constraints**

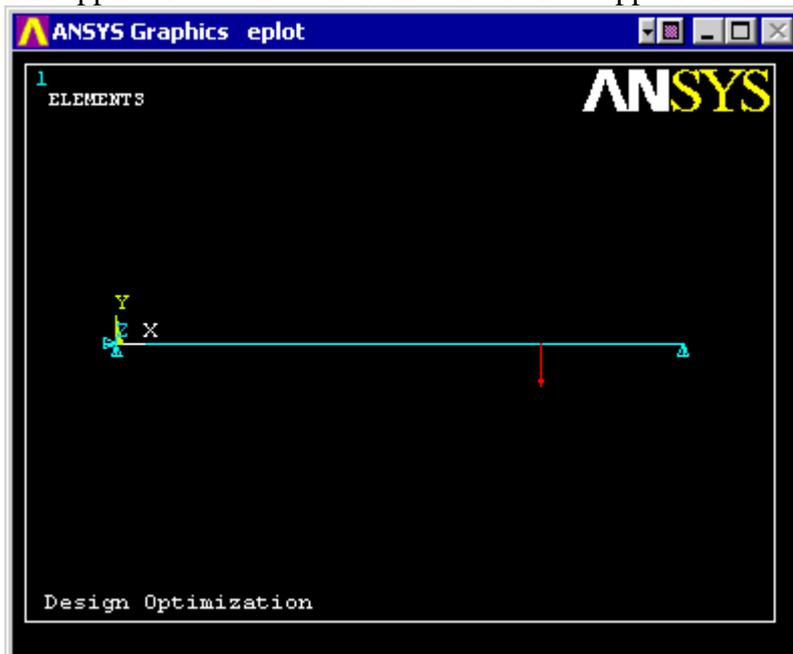   Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

   Pin Keypoint 1 (ie UX, UY constrained) and constrain Keypoint 2 in the Y direction.

3. **Apply Loads**

   Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

   Apply a vertical (FY) point load of -2000N at Keypoint 3

   The applied loads and constraints should now appear as shown in the figure below.

4. **Solve the System**

   Solution > Solve > Current LS

   `SOLVE`

Postprocessing: Viewing the Results

Extracting Information as Parameters:

To perform an optimization, we must extract the required information.

In this problem, we would like to find the maximum stress in the beam and the volume as a result of the width and height variables.

1. **Define the volume**
   - Select **General Postproc > Element Table > Define Table... >** Add...
   - The following window will appear. Fill it in as shown to obtain the volume of the beam.

Note that this is the volume of each element. If you were to list the element table you would get a volume for each element. Therefore, you have to sum the element values together to obtain the total volume of the beam. Follow the instructions below to do this.

o   Select **General Postproc > Element Table > Sum of Each Item...**
o   A little window will appear notifying you that the tabular sum of each element table will be calculated. Click 'OK'

You will obtain a window notifying you that the EVolume is now 400000 mm^2

2.   **Store the data (Volume) as a parameter**

o   Select **Utility Menu > Parameters > Get Scalar Data...**
o   In the window which appears select 'Results Data' and 'Elem table sums'
o   the following window will appear. Select the items shown to store the Volume as a parameter.
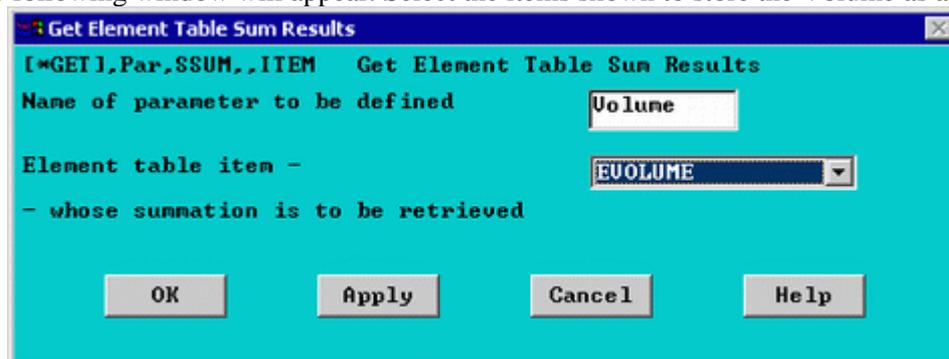
Now if you view the parameters (Utility Menu > Parameters > Scalar Parameters...) you will see that Volume has been added.

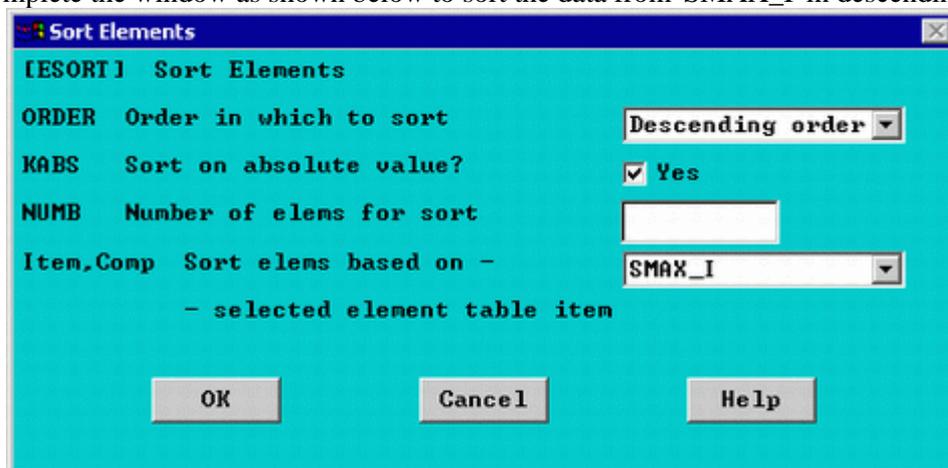3.   **Define the maximum stress at the i node of each element in the beam**

o   Select **General Postproc > Element Table > Define Table... >** Add...
o   The following window will appear. Fill it in as shown to obtain the maximum stress at the i node of each element and store it as 'SMAX_I'.

۴

Note that nmisc,1 is the maximum stress. For further information type `Help beam3` into the command line

Now we will need to sort the stresses in descending order to find the maximum stress

o Select **General Postproc > List Results > Sorted Listing > Sort Elems**

o Complete the window as shown below to sort the data from 'SMAX_I' in descending order

4. **Store the data (Max Stress) as a parameter**
   o Select **Utility Menu > Parameters > Get Scalar Data...**
   o In the window which appears select 'Results Data' and 'Other operations'
   o In the that appears, fill it in as shown to obtain the maximum value.
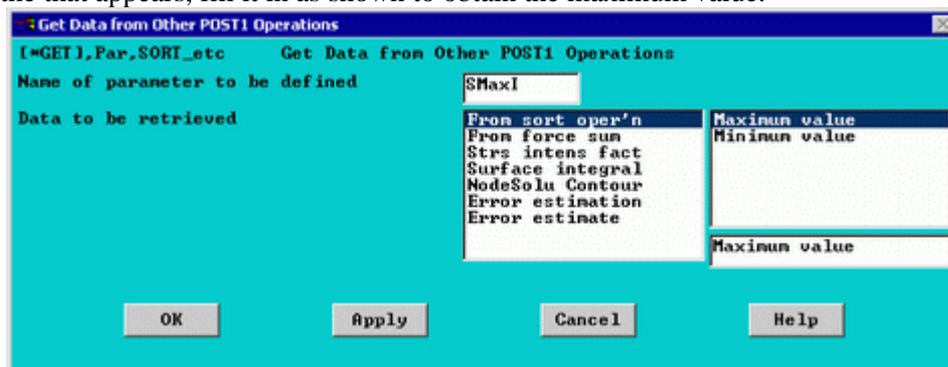
5. **Define maximum stress at the j node of each element for the beam**
   o Select **General Postproc > Element Table > Define Table... > Add...**
   o Fill this table as done previously, however make the following changes:
      ▪ save the data as 'SMAX_J' (instead of 'SMAX_I')
      ▪ The element table data enter NMISC,3 (instead of NMISC,1). This will give you the max stress at the j node.
   o Select **General Postproc > List Results > Sorted Listing > Sort Elems** to sort the stresses in descending order.

- o However, select 'SMAX_J' in the Item, Comp selection box

6. **Store the data (Max Stress) as a parameter**
    - o Select **Utility Menu > Parameters > Get Scalar Data...**
    - o In the window which appears select 'Results Data' and 'Other operations'
    - o In the that appears, fill it in as shown previously , however, name the parameter 'SMaxJ'.

7. **Select the largest of SMAXJ and SMAXI**
    - o Type SMAX=SMAXI>SMAXJ into the command line

        This will set the largest of the 2 values equal to SMAX. In this case the maximum values for each are the same. However, this is not always the case.

8. **View the parametric data**

    Utility Menu > Parameters > Scalar Parameters

    Note that the maximum stress is 281.25 which is much larger than the allowable stress of 200MPa

Design Optimization

Now that we have parametrically set up our problem in ANSYS based on our initial width and height dimensions, we can now solve the optimization problem.

1. **Write the command file**

    It is necessary to write the outline of our problem to an ANSYS command file. This is so that ANSYS can iteratively run solutions to our problem based on different values for the variables that we will define.

    - o Select **Utility Menu > File > Write DB Log File...**
    - o In the window that appears type a name for the command file such as 'optimize.txt'
    - o Click 'OK'.

    If you open the command file in a text editor such as Notepad, it should similar to this:

```
/BATCH
! /COM,ANSYS RELEASE  7.0    UP20021010        16:10:03    05/26/2003
/input,start70,ans,'C:\Program Files\Ansys
Inc\v70\ANSYS\apdl\',,,,,,,,,,,,,,1
/title, Design Optimization
*SET,W , 20
*SET,H , 20
/PREP7
K,1,0,0,,
K,2,1000,0,,
L,      1,      2
!*
HPTCREATE,LINE,1,0,RATI,0.75,
!*
ET,1,BEAM3
!*
!*
R,1,W*H,(W*H**3)/12,H, , , ,
!*
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,200000
MPDATA,PRXY,1,,.3
!*
LESIZE,ALL,100, , , ,1, , ,1,
LMESH,      1
FINISH
/SOL
!*
ANTYPE,0
FLST,2,1,3,ORDE,1
FITEM,2,1
!*
/GO
```

۶

```
DK,P51X, , , ,0,UX,UY, , , , ,
FLST,2,1,3,ORDE,1
FITEM,2,2
!*
/GO
DK,P51X, , , ,0,UY, , , , , ,
FLST,2,1,3,ORDE,1
FITEM,2,3
!*
/GO
FK,P51X,FY,-2000
! /STATUS,SOLU
SOLVE
FINISH
/POST1
AVPRIN,0,0,
ETABLE,EVolume,VOLU,
!*
SSUM
!*
*GET,Volume,SSUM, ,ITEM,EVOLUME
AVPRIN,0,0,
ETABLE,SMax_I,NMISC, 1
!*
ESORT,ETAB,SMAX_I,0,1, ,
!*
*GET,SMaxI,SORT,,MAX
AVPRIN,0,0,
ETABLE,SMax_J,NMISC, 3
!*
ESORT,ETAB,SMAX_J,0,1, ,
!*
*GET,SMaxJ,SORT,,MAX
*SET,SMAX,SMAXI>SMAXJ
! LGWRITE,optimization,,C:\Temp\,COMMENT
```

Several small changes need to be made to this file prior to commencing the optimization. If you created the geometry etc. using command line code, most of these changes will already be made. However, if you used GUI to create this file there are several occasions where you used the graphical picking device. Therefore, the actual items that were chosen need to be entered. The code 'P51X' symbolizes the graphical selection. To modify the file simply open it using notepad and make the required changes. Save and close the file once you have made all of the required changes. The following is a list of the changes which need to be made to this file (which was created using the GUI method)

- Line 32 - `DK,P51X, ,0, ,0,UX,UY, , , , ,`
  Change this to: `DK,1, ,0, ,0,UX,UY,`
  This specifies the constraints at keypoint 1
- Line 37 - `DK,P51X, ,0, ,0,UY, , , , , ,`
  Change to: `DK,2, ,0, ,0,UY,`
  This specifies the constraints at keypoint 2
- Line 42 - `FK,P51X,FY,-2000`
  Change to: `FK,3,FY,-2000`
  This specifies the force applied on the beam

There are also several lines which can be removed from this file. If you are comfortable with command line coding, you should remove the lines which you are certain are not required.

2. **Assign the Command File to the Optimization**
   - Select **Main Menu > Design Opt > Analysis File > Assign**
   - In the file list that appears, select the filename that you created when you wrote the command file.
   - Click 'OK'.

3. **Define Variables and Tolerances**

ANSYS needs to know which variables are critical to the optimization. To define variables, we need to know which variables have an effect on the variable to be minimized. In this example our objective is to minimize the volume of a beam which is directly related to the weight of the beam.

ANSYS categorizes three types of variables for design optimization:

Design Variables (DVs)

Independent variables that directly effect the design objective. In this example, the width and height of the beam are the DVs. Changing either variable has a direct effect on the solution of the problem.

State Variables (SVs)

Dependent variables that change as a result of changing the DVs. These variables are necessary to constrain the design. In this example, the SV is the maximum stress in the beam. Without this SV, our optimization will continue until both the width and height are zero. This would minimize the weight to zero which is not a useful result.

Objective Variable (OV)

The objective variable is the one variable in the optimization that needs to be minimized. In our problem, we will be minimizing the volume of the beam.

NOTE: As previously stated, none of the variables defined in ANSYS are allowed to have negative values.

Now that we have decided our design variables, we need to define ranges and tolerances for each variable. For the width and height, we will select a range of 10 to 50 mm for each. Because a small change in either the width or height has a profound effect on the volume of the beam, we will select a tolerance of 0.01mm. Tolerances are necessary in that they tell ANSYS the largest amount of change that a variable can experience before convergence of the problem.

For the stress variable, we will select a range of 195 to 200 MPa with a tolerance of 0.01MPa.

Because the volume variable is the objective variable, we do not need to define an allowable range. We will set the tolerance to 200mm3. This tolerance was chosen because it is significantly smaller than the initial magnitude of the volume of 400000mm3 (20mm x 20mm x 1000mm).

a. **Define the Design Variables (width and height of beam)**
   - Select **Main Menu > Design Opt > Design Variables... > Add...**
   - Complete the window as shown below to specify the variable limits and tolerances for the height of the beam.
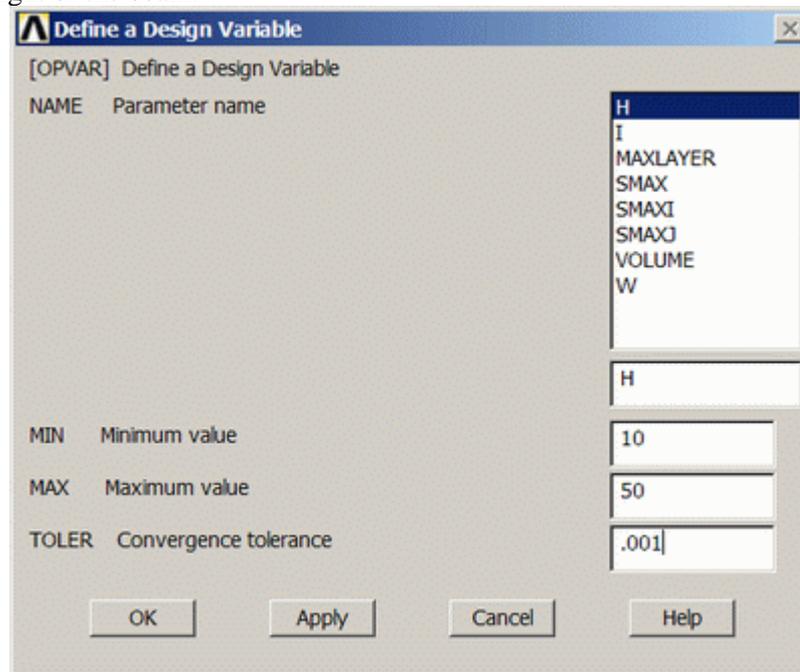
   - Repeat the above steps to specify the variable limits for the width of the beam (identical to specifications for height)

b. **Define the State Variables**
   - Select **Main Menu > Design Opt > State Variables... > Add...**

- In the window fill in the following sections
  - Select 'SMAX' in the 'Parameter Name' section.
  - Enter: Lower Limit (MIN = 195)
  - Upper Limit (MAX = 200)
  - Feasibility Tolerance (TOLER = 0.001)

c. **Define the Objective Variable**
  - Select **Main Menu > Design Opt > Objective...**
  - Select 'VOLUME' in the 'Parameter Name' section.
  - Under Convergence Tolerance, enter 200.

4. **Define the Optimization Method**

There are several different methods that ANSYS can use to solve an optimization problem. To ensure that you are not finding a solution at a local minimum, it is advisable to use different solution methods. If you have trouble with getting a particular problem to converge it would be a good idea to try a different method of solution to see what might be wrong.

For this problem we will use a First-Order Solution method.

  o Select **Main Menu > Design Opt > Method / Tool...**
  o In the 'Specify Optimization Method' window select 'First-Order'
  o Click 'OK'
  o Enter: Maximum iterations (NITR = 30), Percent step size SIZE = 100, Percent forward diff. DELTA = 0.2
  o Click 'OK'.

Note: the significance of the above variables is explained below:

NITR

Max number of iterations. Defaults to 10.

SIZE

% that is applied to the size of each line search step. Defaults to 100%

DELTA

forward difference (%) applied to the design variable range that is used to compute the gradient. Defaults to 0.2%

5. **Run the Optimization**
  o Select **Main Menu > Design Opt > Run...**
  o In the 'Begin Execution of Run' window, confirm that the analysis file, method/type and maximum iterations are correct.
  o Click 'OK'.

The solution of an optimization problem can take awhile before convergence. This problem will take about 15 minutes and run through 19 iterations.

View the Results

1. **View Final Parameters**

Utility Menu > Parameters > Scalar Parameters...

You will probably see that the width=13.24 mm, height=29.16 mm, and the stress is equal to 199.83 MPa with a volume of 386100mm^2.

2. **View graphical results of each variable during the solution**
  o Select **Main Menu > Design Opt > Design Sets > Graphs / Tables...**
  o Complete the window as shown to obtain a graph of the height and width of the beam changing with each iteration
    A. For the 'X-variable parameter' select 'Set number'.
    B. For the 'Y-variable parameter' select 'H' and 'W'.
    C. Ensure that 'Graph' is selected (as opposed to 'List')

۹
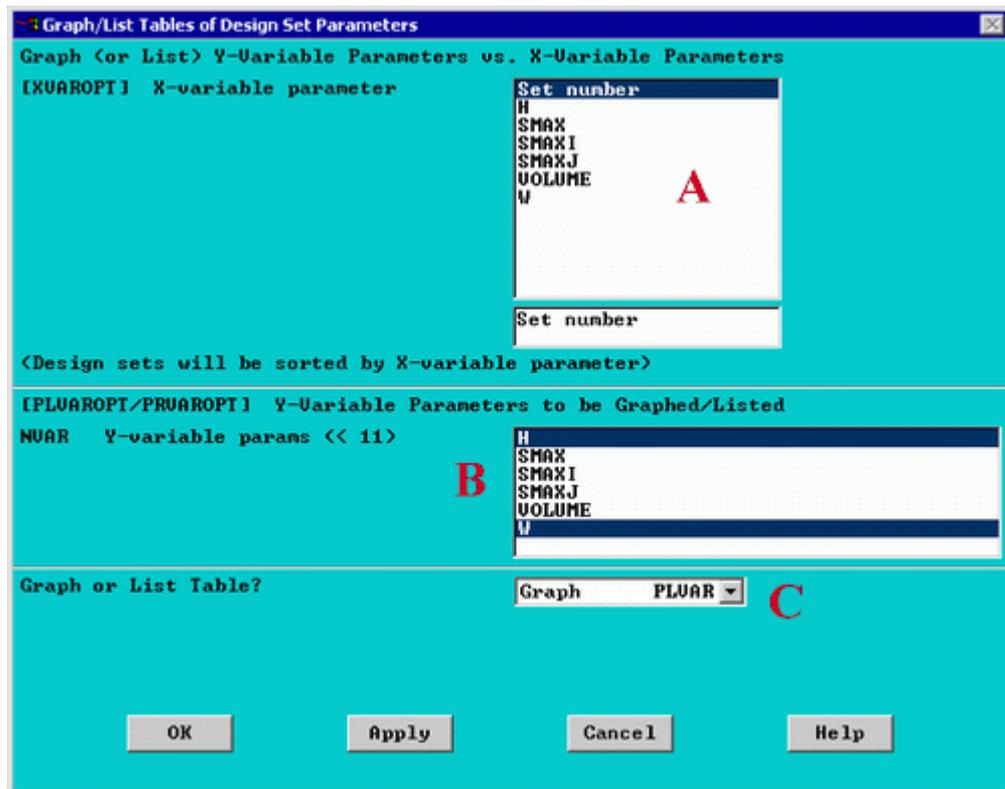
Now you may wish to specify titles for the X and Y axes
- Select **Utility Menu > Plot Ctrls > Style > Graphs > Modify Axes...**
- In the window, enter 'Number of Iterations' for the 'X-axis label' section.
- Enter 'Width and Height (mm)' for the 'Y-axis label'.
- Click 'OK'
- Select **Utility Menu > PlotCtrls**

In the graphics window, you will see a graph of width and height throughout the optimization. You can print the plot by selecting **Utility Menu > PlotCtrls > Hard Copy...**
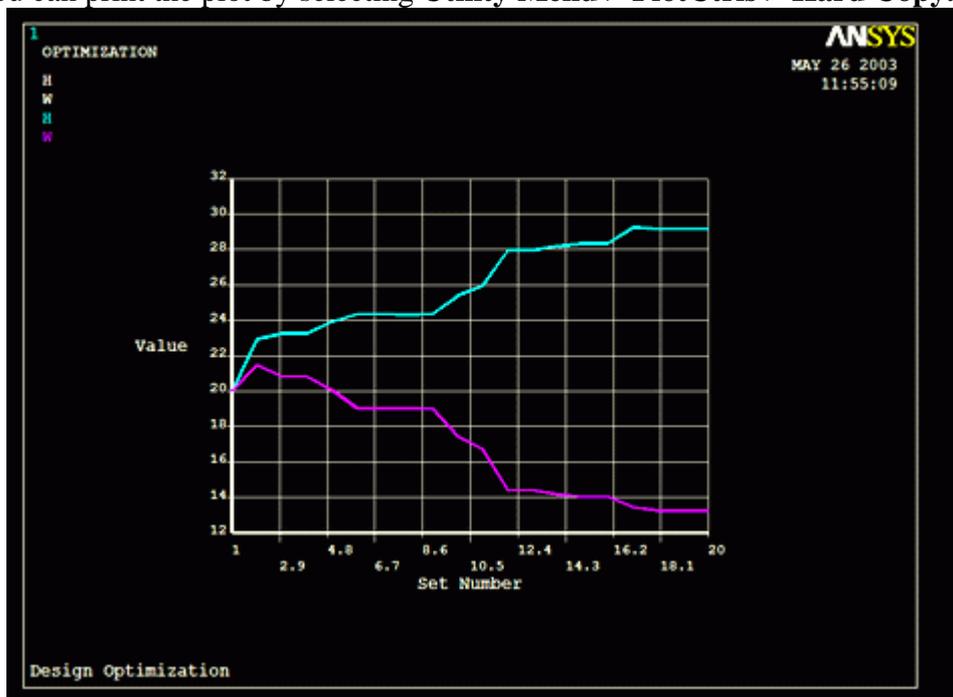
You can plot graphs of the other variables in the design by following the above steps. Instead of using width and height for the y-axis label and variables, use whichever variable is necessary to plot. Alternatively, you could list the data by selecting **Main Menu > Design Opt > Design Sets > List...** . In

١٠

addition, all of the results data (ie stress, displacement, bending moments) are available from the General Postproc menu.

Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Copy and paste following code into Notepad or a similar text editor and save it to your computer. Now go to **'File > Read input from...'** and select the file.

```
/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
node
ESORT,ETAB,SMAX_I,0,1,,
*GET,SMAXI,SORT,,MAX

ETABLE,SMAX_J,NMISC,3                ! Create parameter 'SMaxJ' for max stress at J
node
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
```

١١

```
/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, tolerance 0.001mm
OPVAR,W,DV,10,50,0.001                 ! Width design variable, min 10 mm, max 50 mm,
tolerance 0.001mm
OPVAR,SMAX,SV,195,200,0.001            ! Height state variable, min 195 MPa, max 200
MPa, tolerance 0.001 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
forward difference
OPEXE                                  ! Run optimization

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

Reference

http://www.mece.ualberta.ca/tutorials/ansys