

# Course Project — Design Optimization

---

## Introduction

Design optimization is a technique that seeks to determine an optimum design. By "optimum design", we mean one that meets all specified requirements but with a minimum expense of certain factors such as weight, surface area, volume, stress, cost, etc. In other words, the optimum design is usually one that is as effective as possible.

Virtually any aspect of your design can be optimized: dimensions (such as thickness), shape (such as fillet radii), placement of supports, cost of fabrication, natural frequency, material property, and so on. During the optimization analysis, the program performs a series of analysis-evaluation-modification cycles. That is, an analysis of the initial design is performed, the results are evaluated against specified design criteria, and the design is modified as necessary. This process is repeated until all specified criteria are met.

The following terminology is used in connection with design optimization:

- **Design Variables (DVs)** are independent quantities that are varied in order to achieve the optimum design. Up to 60 DVs may be defined in an ANSYS design optimization problem. Ansys requires a suitable range for each design variable.
- **State Variables (SVs)** are quantities that are used to define constraints for the design. They are also known as dependent variables and are typically response quantities such as stresses and displacements. You may define up to 100 SVs in an ANSYS design optimization problem. A suitable range must be defined for each state variable.
- **Objective Function** is the dependent variable that you are attempting to minimize. It should be a function of the DVs, i.e., changing the value of the DVs should change the value of the objective function. You can define only one objective function in an ANSYS design optimization problem.

In ANSYS, the procedure for design optimization consists of the following main steps:

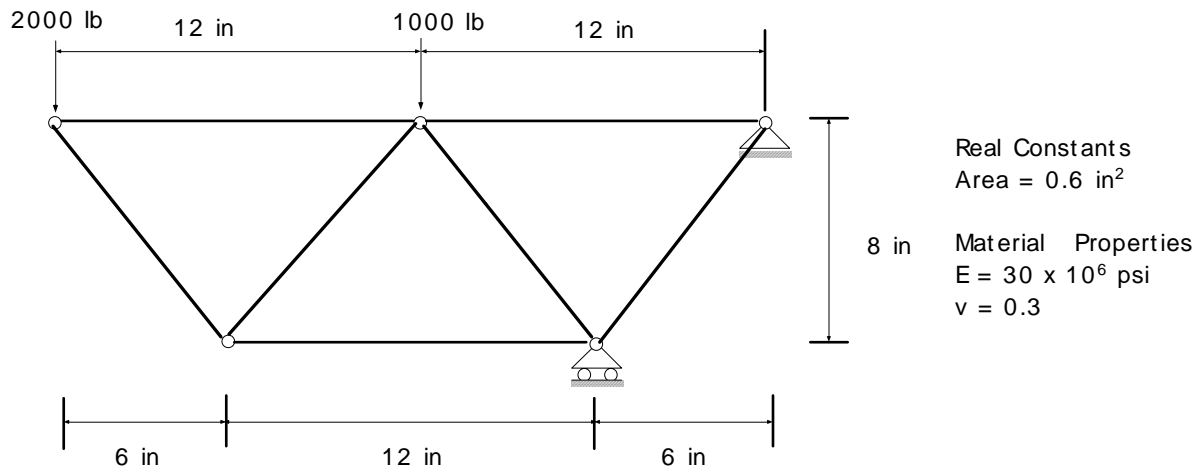
1. Create an analysis file (i.e., a log or a lgw file) to be used during optimization loops. Remember that there are two log files created by ANSYS, one with extension .log, and the other with extension .lgw. The latter is an abbreviated version of the former, and is created upon request. The file should represent a complete analysis, and must contain the following steps:
  - a. Build the model parametrically (PREP7),
  - b. Obtain the solution (SOLUTION),
  - c. Retrieve and assign to parameters the response quantities that will be used as state variables and objective function (POST1)

2. Initialize the optimization analysis.
  - a. Enter OPT and specify the analysis file (OPT).
  - b. Declare optimization variables (OPT)
  - c. Choose optimization method (OPT)
  - d. Specify optimization looping controls (OPT)
3. Carry out the optimization loop (OPT)
4. Review the resulting design sets data (OPT) and post-process results (POST1)

## Example

Find the minimum-weight (i.e., minimum volume) design of the truss structure shown in figure. Treat the cross-sectional area as the design variable and subject the model to the following constraints:

- Stress  $s$  in any member should be in the range  $-20000 < s < 10000$
- Vertical deflection should not exceed 0.05
- Cross sectional area should be in the range  $0.25 < A < 2.5$ .



In this problem, the design variable is the cross sectional area, the *state variables* are the maximum displacements ( $d$ ) and the axial stresses ( $s$ ) in the truss system. The total volume of all elements is the objective function to be minimized.

### 1. Build the model parametrically and initialize design variables

The quantities used for DVs and SVs must be assigned an alphanumeric label and their values initialized using the Parameters menu. The parameters for design variables are defined at the beginning of the model creation. The design variables must be initialized, the initial values can be specified arbitrarily, as far as they are within the acceptable range. The parameters for SVs will be created in the postprocessor after solving the model.

In this example, area of cross-section is the only design variable. Thus only one parameter called AREA is defined here. Later in the input all reference to parameter should be through its name.

- ANSYS Utility Menu > Parameters > Scalar Parameters > Selection [ **AREA=0.6** ]
- Parameters > Save Parameters > OK
- ANSYS Toolbar > SAVE\_DB

If we wanted to use different sections for different members in the truss, we could use several parameters, say AREA1, AREA2, etc. While defining real constants, then we would refer to the appropriate parameters.

## 2. Define the element type (LINK1) and material properties.

- Preprocessor > Element Type > Add/Edit/Delete > Add > Structure Link > [ **2D spar 1** ]
- Preprocessor > Real Constants > Add/Edit/Delete > Add > Real Constant Set No.[1] > Cross Sec Area [ **AREA** ]
- Preprocessor > Material Props > Material Models > Structural > Linear Elastic > Isotropic >> EX [ **30E6** ] > PRXY [ **0.3** ]

## 3. Begin the model creation.

- Preprocessor > -Modeling- **Create**  
Nodes > In Active CS  
NODE [ **1** ] X,Y,Z [ **6** ] [ **0** ] [ **0** ] > Apply  
NODE [ **2** ] X,Y,Z [ **18** ] [ **0** ] [ **0** ] > Apply  
NODE [ **3** ] X,Y,Z [ **0** ] [ **8** ] [ **0** ] >Apply  
NODE [ **4** ] X,Y,Z [ **12** ] [ **8** ] [ **0** ] >Apply  
NODE [ **5** ] X,Y,Z [ **24** ] [ **8** ] [ **0** ] >OK
- Preprocessor > -Modeling- **Create**  
Elements > -Auto Numbered- **Thru Nodes** +  
[ Node 1 and 2 ] > Apply  
[ Node 1 and 3 ] > Apply  
[ Node 1 and 4 ] > Apply  
[ Node 2 and 4 ] > Apply  
[ Node 2 and 5 ] > Apply  
[ Node 3 and 4 ] > Apply  
[ Node 4 and 5 ] > OK

## 4. Apply boundary conditions and loading.

- Solution > -Define Loads- **Apply** > -Structural- **Displacements** > On nodes

[ Node 5]: All dof  
[ Node 2]: UY

- Solution > -Define Loads- **Apply** > -Structural- **Force/Moments** > On nodes

- Apply loads  
[ Node 4]: [ **FY** ] > VALUE [ **-1000** ]  
[ Node 3]: [ **FY** ] > VALUE [ **-2000** ]

## 5. Initiate the solution

- Solution > -Solve- **Current LS**

## 6. Retrieve results parametrically in the postprocessor.

This is where you retrieve results data and assign them to parameters. ANSYS provides a number of commands that allows us to retrieve the value of selected variables, to pick out the maximum/minimum value from a pre-selected result array, and so on. At the end of an

optimization loops, the state variables are evaluated, and the design variable are reset to continue the analysis.

Important Note: Follow the steps below very carefully. Make sure to click on options in the dialog boxes even if they appear already selected. If you don't do this Ansys sometimes does not create a proper LGW file needed for optimization.

- General Postproc > Read Result > Last Set

We sort nodes based on absolute value of UY deflection.

- General Postproc > List Results > -Sorted Listing- **Sort Nodes...** > KABS [ **Yes** ]  
Item, Comp [ **DOF solution** ] [ **Translation UY** ] > OK

We now define a parameter DMAX = maximum absolute UY deflection

- ANSYS Utility Menu > Parameters >  
Get Scalar Data > [ **Result data** ] [ **Other operations** ] > OK  
Name of parameter to be defined [ **DMAX** ]  
Data to be retrieved [ **From sort oper'n** ] [ **Maximum value** ] > OK

Next we set-up element table to extract volume and axial stress for each element. The command VOLU is needed to get the element volume and LS, 1 is needed to get element stress. The labels AXIAL and VOL are user-specified names for the element volume and axial stress.

- General Postproc > Element Table > Define Table > Add... > Lab [ **VOL** ]  
[ **Geometry** ] [ **Elem volume VOLU** ] > OK > Close
- General Postproc > Element Table > Sum of Each Item > OK
- General Postproc > Element Table > Define Table > Add... >  
Lab [ **AXIAL** ] [ **By sequence num** ] [ **LS** ] > Selection [ **LS, 1** ] > OK > Close

Now we can use the element table data to create three remaining SVs, VOLUME = total volume of elements, SMAX = maximum axial stress and SMIN= minimum axial stress. For volume we need to choose element table sums so that we can get total volume of the truss and use it in defining the objective function.

- Parameters > Get Scalar Data > [ **Result data** ] [ **Elem table sums** ] > OK > Name of parameter to be defined [ **VOLUME** ] Element table item [ **VOL** ] > OK
- General Postproc > List Results > -Sorted Listing- **Sort Elems...** > KABS [ **no** ]  
Item, Comp    [ **AXIAL** ] > OK
- Parameters > Get Scalar Data > [ **Result data** ] [ **Other operations** ] > OK > Name of parameter to be defined [ **SMAX** ] Data to be retrieved [ **From sort oper'n** ] [ **Maximum value** ] > Apply  
[ **Result data** ] [ **Other operations** ] > OK > Name of parameter to be defined [ **SMIN** ] Data to be retrieved [ **From sort oper'n** ] [ **Minimum value** ] > OK

## 7. Create the analysis file -- Important Step!

The following command creates a file (*jobname.lgw*) that represents the model database. This file contains all commands that were used to create the current database. Ansys will use this file repeatedly during optimization iterations. Therefore it is important to make sure that this file is free of any errors.

- ANSYS Utility Menu > File > **write DB Log File** > OK

## 8. Initiate optimization analysis

Assign the *jobname.LGW* as the optimization analysis file.

- Design Opt > -Analysis File- **Assign...** > **jobname.lgw** > OK

Choose design and state variables from the list of parameters created earlier and define appropriate ranges.

- Design Opt > Design Variables... > Add...

>   **AREA** MIN [ **0.25** ] MAX [ **2.5** ] > OK > Close

- Design Opt > State Variables... > Add...

>   **DMAX** MIN [ **0** ] MAX [ **0.05** ] > Apply

>   **SMAX** MIN [ **-20000** ] MAX [ **10000** ] > Apply

>   **SMIN** MIN [ **-20000** ] MAX [ **10000** ] > OK > Close

Choose objective function from the list of parameters created earlier.

- Design Opt > Objective... >   **VOLUME** > OK

Choose an optimization method. Select the maximum number of iterations.

- Design Opt > Method/Tool... > MNAME   [ **Sub-Problem** ] > OK  
> NITR [ **30** ] NINFS [ **7** ] > OK

Note. ANSYS offers two optimization methods: the sub-problem approximation method and the first order method. The first method essentially sweeps through the values of the design variables in the ranges and selects the optimal design. The second involves more sophisticated mathematical tools (e.g., sensitivity analysis). In either case, the solution is obtained through a number of iterations.

Start the optimization iterations

- Design Opt > Run... > Begin Execution of Run > OK

ANSYS will stop if it finds an optimum or if the number of iterations reaches the specified maximum. Once the loop terminates, we can then review results and decide whether further iterations are needed. There are options for plotting history of objective function and other parameters as a function of number of iterations.

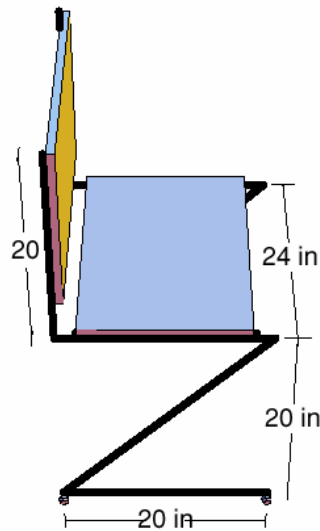
- Design Opt > -Design Sets- **List...** > OK

- Design Opt > -Design Sets- **Graphs/Tables** > [XVAROPT]   Set number  
NVAR   **VOLUME** > OK

## Project: Design Optimization

An office furniture company is designing a new low cost chair. The basic concept calls for two upholstered cushions over sturdy plywood boards that are supported by two Z-shaped frames. Your assignment is to come up with the lowest cost design for the frame while meeting the following requirements:

- The frames must be able to support a weight of 300 lb from the seat.
- To simulate the forces from a person sitting while leaning against the back, the frames must be able to support a vertical load of 250 lb from the seat and horizontal load of 50 lb from the back.
- Any part of the frame should not deflect more than 1/8 in.
- The factor of safety against stress failure should be at least 3.



1. Consider the frame to be made of 2 in diameter steel tube with wall thickness 0.25 in. Model the frame with **beam elements**, create a finite element mesh that contains few tens of elements. Determine if the given design meets the stress and the deflection constraints. For simplicity, assume the loads exerted from the plywood boards are uniformly distributed. What is the weight of the frame in this design?

2. Use design optimization capability of ANSYS to obtain a minimum weight design. For ease in manufacturing the same diameter must be used for the entire frame. However, the wall-thickness can be different for various part of the frame. Don't consider the weight of the plywood boards.

Use the following material parameters:  $E = 30 \times 10^6$  lb/in<sup>2</sup>, failure stress  $\sigma_y = 60000$  lb/in<sup>2</sup> and weight density  $\rho = 0.256$  lb/in<sup>3</sup>.