UNIVERSIDAD DE SAN BUENAVENTURA
FACULTAD DE INGENIERÍA
PROGRAMA DE INGENIERÍA MECATRÓNICA

MECHANICS OF
MATERIALS
2ND TERM
2021-II

Author:
Nikolay Prieto Ph.D(c)

Computational Laboratory ANSYS (Part I)

Contenido

1. Introduction

Discrete mathematics techniques allowed to approximate physical mechanical environments into computational models to find the reactions to those boundary conditions. The finite element method requires the system geometry to be defined by a number of points in space called nodes. Each node has a set of degrees of freedom (temperature, displacements, etc.) that can vary based on the inputs to the system. These nodes are connected by elements that define the mathematical interactions of the degrees of freedom (DOFs).

1.1. Basic procedure for Finite element Analysis

There are 10 basic steps in any FEA. First, the solid model geometry is created, the element type(s) and material properties are defined, and the solid model geometry is meshed to create the finite element model. In ANSYS, these steps are performed in the Preprocessor (PREP). Next, loads and constraints are applied, solution options are defined, and the problem is solved. These steps are performed in the solution processor (SOL). After the solution is ready, the results are plotted, viewed, and exported in one of the postprocessors (POST). Finally, the results are compared to first-order estimates, closed-form solutions, mathematical models, or experimental results to ensure that the output of the program is reasonable and as expected

- Define the solid model geometry *(PREP)*

Universidad de San Buenaventura
Facultad de Ingeniería
Programa de Ingeniería Mecatrónica

Mechanics of
Materials
2nd Term
2021-II

- Select the element types. *(PREP)*

- Define the material Properties.*(PREP)*

- Mesh. *(PREP)*

- Define the boundary conditions. *(SOL)*

- Define the loads *(SOL)*

- Set the solution options *(SOL)*

- Solve *(SOL)*

- Plot, view, and Export the results *(POST)*

- Compare and Verify the results *(POST)*

1.2. ANSYS is an Engineering Software, not an engineer!

As with all computer programs, the quality of your results will depend on the quality of your model. This includes the accuracy of the material properties, the appropriateness of the material models, how closely the simulated geometry and loads match the actual geometry and loads, and the validity of the simplifications and assumptions made. In general terms, Garbage In – Garbage Out. Finite element software programs can be thought of as very sophisticated calculators that help you to analyze engineering systems that could not otherwise be evaluated. They integrate the section properties of the system with the material properties to generate the equations to be solved. They convert the applied loads to the appropriate forms and apply them to the specified DOFs. They solve the generated system of equations. And, they help you to visualize and understand the results. But a finite element program will not comment on the validity of any assumptions made in setting up the model as long as the laws of physics are not violated. It also will not ensure that you are using the correct laws of physics for a given problem. Any errors that the program reports will be associated with the use of the program, and not with the physical or analytical system. In addition, it will not provide any commentary on the quality or implications of the results. Finite element software is only a tool. In the end, you, and you alone, are responsible for determining whether or not the results of your finite element model can be used to make or justify engineering decisions.

1.3. Physics Capabilities

Structural analyses may include linear and nonlinear buckling; fracture; composites; fatigue; and contact with and without friction, gaskets, joints, pretension, and spot welds. They can involve geometric non-linearities such as large strain and large deflection, and may use linear and non-linear material models including rate-dependent and rate-independent plasticity, hyper-elasticity, visco-elasticity, and creep. Structural analyses where time is important but time steps are very short, such as high-speed impacts and explosions, should be performed using one of the ANSYS explicit dynamics products such as ANSYS LS-DYNA or ANSYS AUTODYN.

2. Static Axial Loading of a Notched plate in tension.

In this exercise, you will perform a steady-state structural analysis of a notched rectangular plate that is loaded axially in tension. The plate is made of 6061-T6 aluminum with a Young's modulus of 73.1 GPa and a

UNIVERSIDAD DE SAN BUENAVENTURA
FACULTAD DE INGENIERÍA
PROGRAMA DE INGENIERÍA MECATRÓNICA

MECHANICS OF
MATERIALS
2ND TERM
2021-II

Poisson's ratio of 0.33. It is 15 cm wide and 10 cm tall with 1 cm diameter notches on each side. The notches are centered horizontally on the plate midplane and centered vertically on the upper and lower plate edges (Figure 2-1-1). The plate is thin so this can be treated as a two-dimensional (2D) problem. The geometry will be created using top down solid modeling techniques including Boolean operations.

A load of 1 MPa will be placed on the right edge of the plate. The left edge of the plate will be constrained in x. This constraint will create a reaction force that will act as the second 1 MPa load on the left. The plate will deform under the applied load, growing longer and thinner. This will force the notches to become ovals and a stress concentration will appear near the center of each notch. The ratio of the height of the plate (10 cm) to the diameter of the notches (1 cm) is 10 to 1. Thus, the expected elastic stress concentration $Kt = \frac{\sigma_{máx}}{\sigma_{nom}}$ for this problem is 2.735 and the expected gross stress concentration $Kg = \frac{\sigma_{máx}}{\sigma_{gross}}$ for this problem is 3.03.
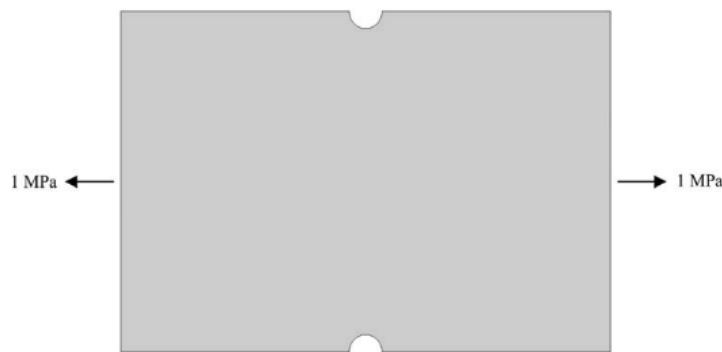

Figura 1: Schematic view of the Notched plate under static conditions.

2.1. Model Attributes

- Young's Modulus 7.310×10^{10} Pa

- Poisson's ratio $v = 0.33$

- Load 1 MPa on the right edge of the plate

- Constrain 1: No displacement on the left edge in horizontal direction (x)

- Constrain 2: No displacement of the lower left corner of the plate in the vertical (y) direction.

3. Structural Analysis of a simple Warren Truss Using Direct Generation

In this exercise, you will create a finite element model of a simple Warren truss using direct generation of the nodes and elements. The truss is 20 in. wide and 10 in. tall. Each member has a 1x1 in. cross section (Figure 2). The model will be built using the US customary system. The truss is made of aluminum 6061-O with a Youg Modulus $E = 1 \times 10^7$ ψ and a Poisson's ratio of 0.33.

The truss will be modeled using spar (truss) elements. This allows uniaxial tension and compression within the members, but no bending of the members. All joints are pinned and can rotate freely. The pin in the lower

Universidad de San Buenaventura
Facultad de Ingeniería
Programa de Ingeniería Mecatrónica

Mechanics of
Materials
2nd Term
2021-II

left corner of the truss is fixed in space. The lower right corner of the truss is supported by rollers. A downward force of 1000 lb-f is applied to the bottom center joint of the truss. Because there are no out-of-plane boundary conditions, the truss will be modeled in 2D. Given these assumptions and boundary conditions, the truss is statically determinate. All displacements and forces can be calculated by hand. Selected results are calculated at the end of the exercise for verification.
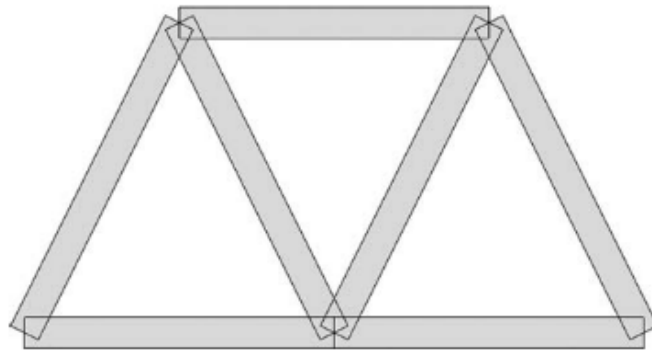

Figura 2: Schematic view of the Warren truss.

3.1. Model Attributes

- Young's Modulus 1×10^7 ψ

- Poisson's ratio $v = 0.33$

- Load 1000 lb-f downward load on the lower center joint Constraints.

- Constrain 1: No displacement on the lower right corner of the truss.

- Constrain 2: No displacement on the lower left corner of the truss in the vertical direction.

- Constrain 3: No out-of-plane displacements

4. Activity

Consider the truss structure shown in Fig. 3. Constrains in x and y are presented in the diagram, The force application point is applied at the bottom right side downwards.

4.1. Model Attributes

- Young's Modulus 200 GPa

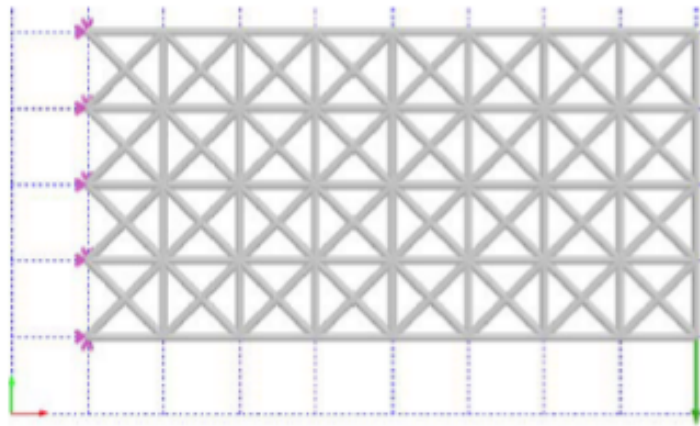- Material: Steel

- Poisson's ratio $v = 0.33$

Universidad de San Buenaventura
Facultad de Ingeniería
Programa de Ingeniería Mecatrónica

Mechanics of
Materials
2nd Term
2021-II

Figura 3: Schematic view of one proposed design.

- The minimum Cross-sectional area of each truss is 25mm × 25mm, however, you can modify this area in whatever truss of the structure so that obtain the lightest weight as possible.

- Consider a maximal length and height of 3m × 1.5m

- Load 2 kN downward load on the bottom right joint.

- Constrain 1: No displacement on the the horizontal axis(x) on the lower left line nodes.

- Constrain 2: No displacement on the lower left line of the truss in the vertical direction.

- Constrain 3: No out-of-plane displacements

- Consider a Factor of Safety $FS = 2.0$

Your task is to find a combination so that the truss is the lightest in consideration of the max stress obtained (including FS). An example is given in Fig. 4

4.2. Assignment requirements

- The work is by pairs

- Write a formal report i.e latex, word, Google Office, etc.

- Establish the methods step by step.

- Write your results in plots, short tables and short paragraphs.

- Write the final weight and stress of the design given. Propose at least three options.

- If you accomplish the previous requirements, your grade will be assign according to the minimum weight overall.

Universidad de San Buenaventura
Facultad de Ingeniería
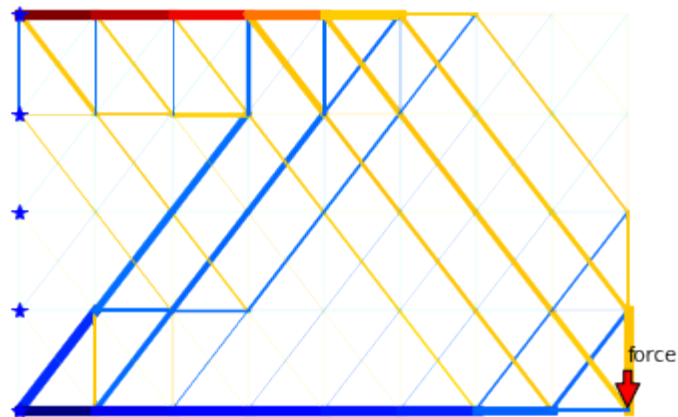Programa de Ingeniería Mecatrónica

Mechanics of
Materials
2nd Term
2021-II

Figura 4: Schematic view of the proposed structure.