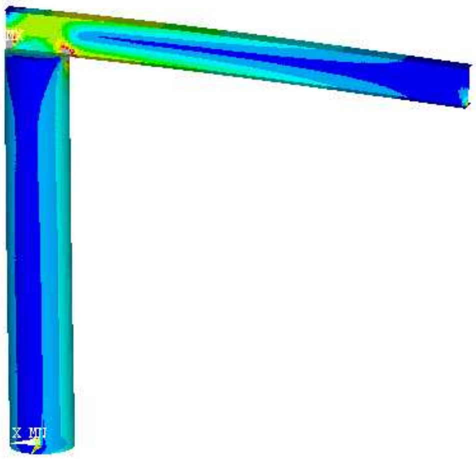
FINITE ELEMENT CRANE ANALYSIS ACCORDING TO UNE 58132-2 STANDARD

MECHANICAL ENGINEERING DEPARTMENT
INDUSTRIAL ENGINEERING



Vicente Díaz López
Antonio Gauchía Babé
Beatriz López Boada
Maria Jesús López Boada
Carolina Álvarez Caldas

1.- OBJECTIVE

The objective of this lab is to analyse, by means of a finite element package (Ansys), a jib crane structure. Static and modal analysis will be carried out. This lab will allow to learn not only Ansys, but the general concepts of the finite element method. In addition, the crane structure analysis will be carried out according to UNE 58132-2 standard.

2.- INTRODUCTION

Finite element methods are currently being applied to solve many mechanical problems. It allows to analyse the crane structure stress, forces, accelerations,... so as to verify that it has been correctly designed. Its main advantage is that no real crane is being subjected to an experimental test. However, simulation results have to be carefully analysed. As an engineer, you are expected to know if the simulation results are correct.

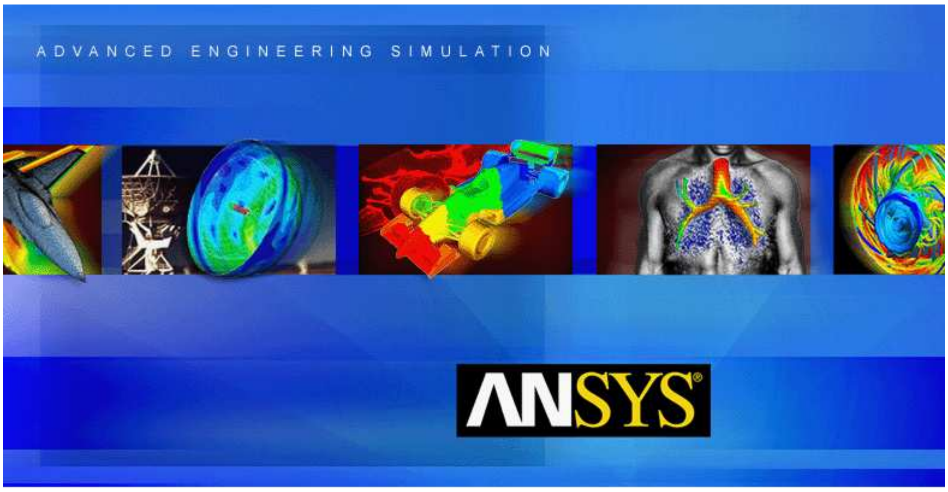


Figure 1. Ansys, a finite element software

3.- BEGINNIG WITH ANSYS

To begin, create a folder wherever you want. Open Ansys and go to File ->Change Directory and select the recent created folder.

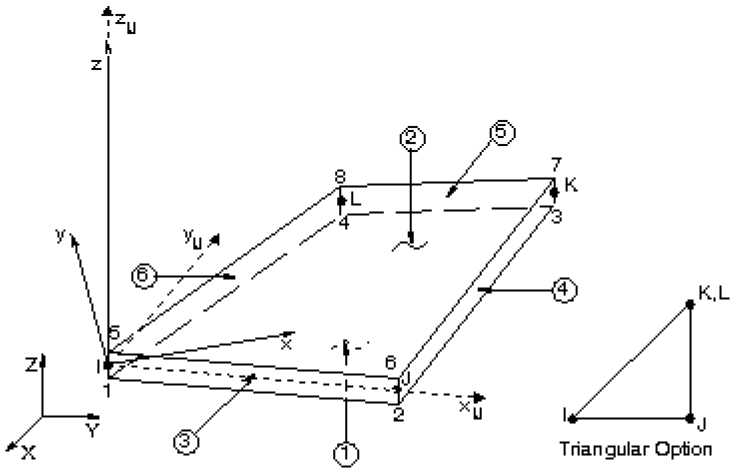
Ansys creates various files, however the main one is .db which generally has a large size. It also creates an .err results where warnings and errors can be reviewed.

To save your results, go to File -> Save as … When you quit the program remember to save only geometry due to the fact that the size of the files is quite large.

4.- PREPROCESSOR

In the first place, geometry will be created by means of points, called keypoints, and lines. Afterwards, it will be selected a type of element and material properties will be defined. Finally, the crane will be meshed.

Finite element method solves a continuous problem (differential equations) by solving a discrete problem (set of algebraic equations). Each element is formed by nodes. Values of force, stress or strain for points between the nodes is calculated by means of the values of force, stress or strain of the nodes using a shape function. Therefore, a shape function is a mathematical equation that establishes a mathematical relationship between the displacement at nodes and the mid points. Depending on the chosen element, the shape function is different. In Figure 2 it shown the shape function of a shell element. The selection of the type of element is of paramount importance and time cost or precision are some of the variables that have to be taken into account when choosing an element.



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

Figure2. Shell element

4.1- Type of element

The element to be selected is Shell 63. The selected element is defined by four nodes I, J, K and L. To select the element go to Preprocessor ->Element Type -> Add/Edit/Delete. Choose Shell 63. Afterwards, real constants have to be defined, go to Real constants -> Add -> and type the thickness value: 25 mm

4.2- Material properties

The material of the crane structure is steel with:

- Modulus of Elasticity: 200 GPa
- Poisson ratio: 0.3
- Density: 7850 kg/m^3

To specify material properties, go to Material Props -> Material Models -> Structural -> Linear -> Elastic -> Isotropic. Ansys does not use any units, that is, depending on the units one wants to specify geometry and the units one wants stress and displacement output to be, material properties values have to be introduced.

4.3- Geometry: Modelling

In figure 3 the crane that will be analysed is depicted.

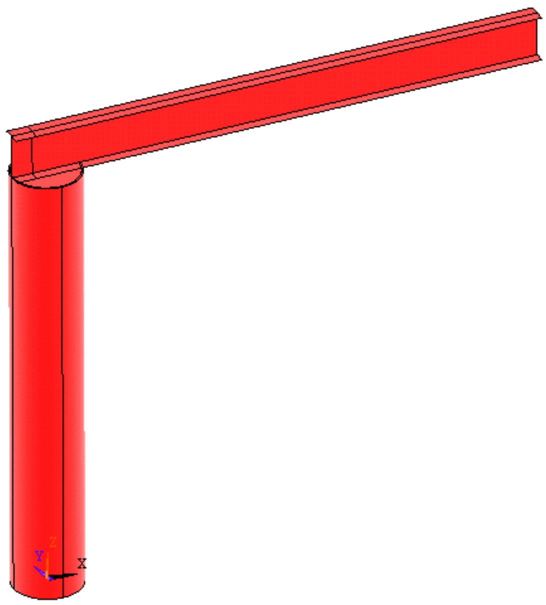


Figura 3.Crane structure

To model the structure, keypoints and lines will be used. To create the keypoints, go to Modelling -> Create -> Keypoints in Active CS. Leave the Keypoint number in blank, so that Ansys will fill it in automatically. Afterwards, fill in the x, y and z coordinates. The best way to do it is to align the mast with the Y direction.

Lines connect the keypoints. To create them pick Modelling -> Create -> Lines -> Lines -> Straight lines. A window is prompted and the mouse turns onto an arrow. To create a line between two keypoints select the first keypoint with the left mouse button and leave it free, afterwards, select the next keypoint. When finished, click OK. By the same procedure, areas can be created: Modelling -> Create -> Areas -> Arbitrary -> By Lines. It is very important to take into account that the dimensions to be introduced in the program are not the real ones. This is due to the fact that, because of the meshing (giving thickness to the structure) the real dimensions are achieved. The meshing, distributes the thickness 50% to each part and therefore, the finite element model is called half plane. For simplicity, a local coordinate system ("Working Plane") can be defined, allowing to define points in an easier way. Go to Workplane -> Offset WP to-> Keypoint and select the keypoint that you wish to select as local coordinate system. Afterwards, go to Worplane -> Change Active CS to -> Working Plane.

1. Mast

To create the mast go to Modelling -> Create -> Lines -> Arcs ->By centre and radius Introduce the value of 0,0 and click OK and afterwards, introduce the value of the mast radius: 500 mm and introduce 360 degrees and the number of lines to define the arc 4. Furthermore, the mast height must be defined. Firstly, introduce the coordinate value of two keypoints that will define the height so that the basement of the mast will be extruded along that line. Therefore, create a keypoint in the global origin and another one at (0,0,6300) because the mast has a height of 6300 mm. Create a line between these two keypoints. Furthermore, click on: Modelling -> Operate -> Extrude -> Lines -> Along lines and select the circle click ok and select the line that defines the height of the mast and click ok.

2. Jib

The jib has a cross section in I of 300 mm by 650 mm and a thickness of 25 mm. It has to be taken into account that the cross section has to be modelled according to the half plane model. Therefore, in the first place, the lines have to be defined and afterwards, the lines will be extruded along a line that defines the length of the jib. When the created areas are meshed, the cross section will be like that depicted in figure 4.

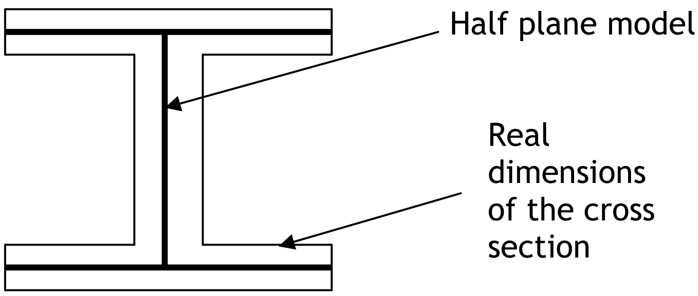


Figure 4.Half plane model

Applying the half plane model, the horizontal part will be 300 mm, however, the central structure of the beam will have a length of 625 mm. In addition, due to the fact that the mast is hollow a circular area has to be defined at its top. To create the areas: Modelling -> Create -> Areas ->Arbitrary -> By lines and select the top circle.

Afterwards, situate the coordinate origin in the top part of the mast and in one of the diametric points of the mast. Plot the lines (view figure 5) that define the bottom part of the jib, the vertical and the top part, knowing that the jib has a total length of 7500 mm.

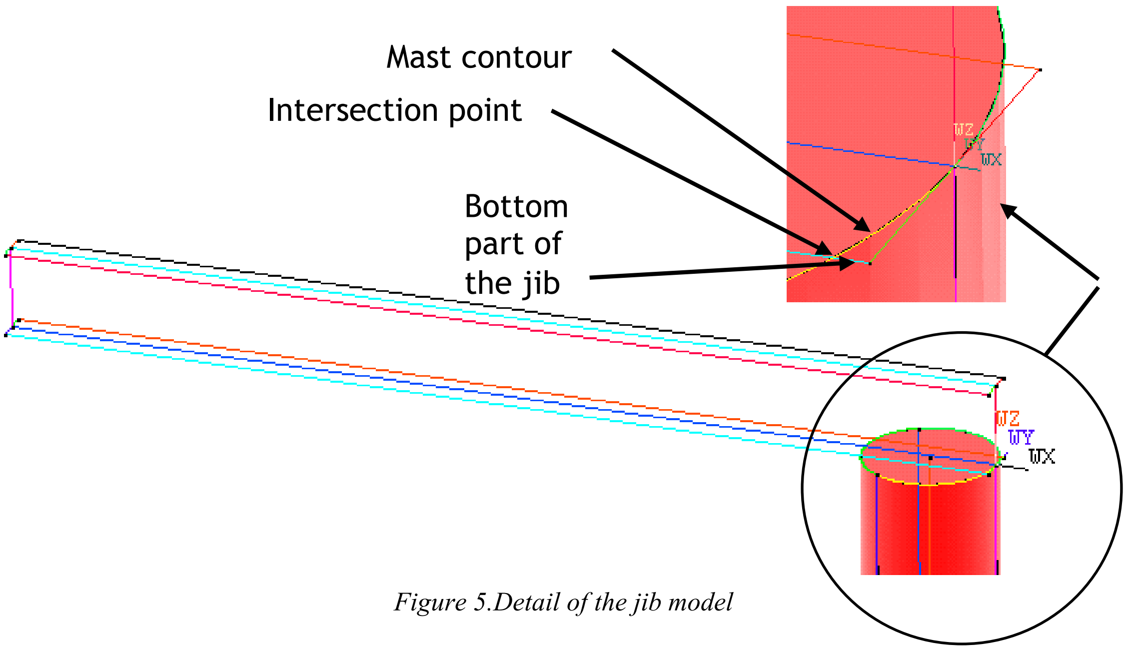


Figure 5.Detail of the jib model

Due to the fact that, the bottom part of the jib does not stick out of the mast the intersection points must be calculated (view figure 5). Click on Modelling -> Operate -> Booleans -> Divide Area by line and select the tope circular area of the mast and click ok and select the three lines. The result is shown in figure 6.

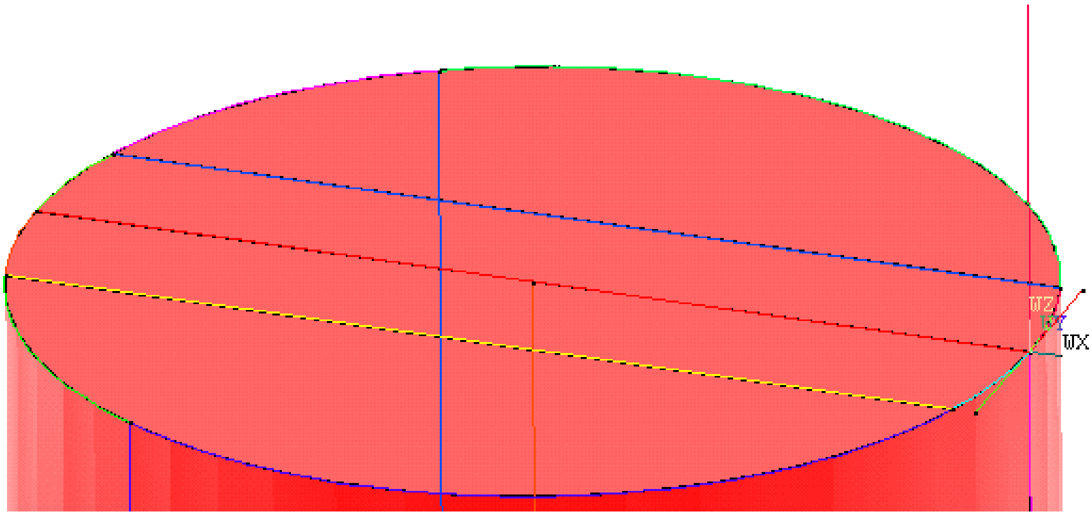


Figure 6.Area division

In figure 6, it can be observed that the program has deleted certain lines of the jib, create them again. Furthermore, create the areas of the bottom part of the jib. It has to be taken into account that the lower part of the jib situated on the mast is already created, and therefore, only the cantilever part has to be defined. Finally, create the central area of the jib.

4.5- Meshing

The created areas have to be meshed. Firstly, the thickness has be defined. Go to Mesh Attributes -> All Areas and introduce the values:

- Material Number: 1
- Real constant set number: Set 1
- Element type number: Shell 63
- Element section: Non defined
- Pick Orientation Keypoints: No

Now the element dimensions have to be defined. Click on Meshing -> Mesh Tool and in Global Set introduce the value of 50 mm. Click Mesh. In figure 7 the meshed crane is shown.

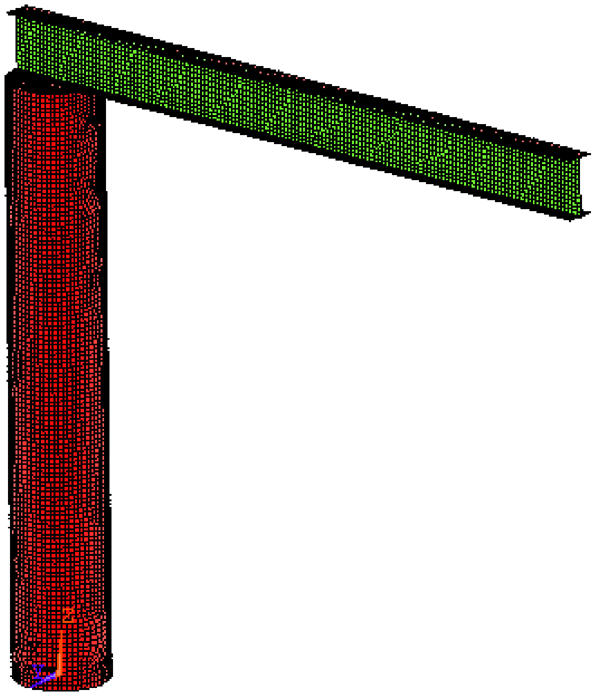


Figure 7.Meshed crane

5.- SOLUTION

In the first place, the type of analysis has to be specified: Analysis Type -> New Analysis -> Static. To apply loads:

- Gravity: Define loads -> Apply -> Structural -> Inertia -> Gravity in the z direction and with positive value.
- Vertical load: Define loads -> Apply -> Structural -> Force/Moment -> Nodes and select the node .
- To apply a pressure (Force per unit area): Define loads -> Apply -> Structural Pressure-> On Areas. The value of the force per unit projected area has to be introduced.
- To apply boundary conditions: Define loads -> Apply -> Structural Displacement-> On Lines

The forces are defined as positive when they go in the positive direction of the coordinate system, however, to check pressure direction you can view the applied pressure by showing it as arrows Plot Controls ->Symbols-> Surface Load symbols -> Show press and convect as Arrows and the click on Plot -> Areas

To delete a certain load case: Define loads -> Delete -> All load data. Finally, to solve the model click on Solve -> Current LS.

6.- GENERAL POSTPROCESSOR

It is of paramount importance to check the correctness of the created model by, for example, computing the reactions loads. Go to List Results -> Reaction Solutions -> All item. Maximum displacement is usually restricted. Go to List Results -> Nodal Solution -> DOF solution -> Translation -> UY and search for the maximum displacement.

Finally, maximum stress can be plotted Plot results -> Contour plot -> Element results and select the stress you want to plot. In the screen you can view the maximum plotted stress (SMX) and its location (MX).

7.- MODAL ANALYSIS

Finally a modal analysis of the jib crane structure will be carried out. Select a new analysis -> Modal Analysis. In Solution -> Analysis Type -> Analysis Options Specify the number of modes to extract, for example, 20, and the frequency range (0 to 100 Hz). Go to Define Loads -> Delete -> All Loads and Options so that no loads will be applied. And solve the model.

Natural vibration frequencies and modes can be calculated. Go to General Postproc. -> Results Summary to list the natural vibration frequencies in Hz. Furthermore, for each of the natural vibration frequencies modes can be animated. Go to General Postproc. -> Read Results -> By Pick. Select one of the frequencies and click on Read. Afterwards, go to the upper menu and click on the Plot Controls -> Animate -> Mode Shape, choose time delay 0.1 s DOF Solution -> Deformed Shape.

8.- CRANE STRUCTURE CALCULATION

To carry out the model and apply the loads, the loads have to be calculated. The crane will have to lift a load of 54900 N. Knowing that the lift speed is 0.08 m/s and the wind speed is 20 m/s, the factors have to be calculated.

9.- WORK

a) For the given wind speed, calculate the wind force that will be applied in the mast and the jib.

b) List the reaction for the two considered cases: Case I (with wind) and Case II (without wind) applying the factors.

c) Calculate the jib weight.

d) Maximum displacement of the jib for case II, taking into account crane self weight and the load. In addition, demonstrate with hand calculations that the value given by ansys is correct. ¿Why are there differences between both methods?

e) Plot the vertical displacement of the crane and the horizontal displacement for the two cases.

f) Maximum stress of the crane and its location for both cases ¿Does it make sense the location of the maximum stress? Justify your answer.

g) For the jib, calculate by means of hand calculations according to UNE 58132-2 standard: the stress (normal stress and tangential stress) and plot the flexure and tangential diagrams due to vertical forces (self weight and load of 54900 N) for case I. Compare the results with the result given by ansys.

h) ¿How would you improve the crane design?

i) ¿Why is modal analysis so important? ¿For what type of cranes is important to carry out a modal analysis?

j) Calculate the 10 first natural vibration frequencies different from zero.

k) ¿If no boundary conditions are applied to the crane, why the first 6 natural vibration frequencies are zero?

l) For the first 10 frequencies plot the modes and specify which one is flexure, torsion or combination of both of them.