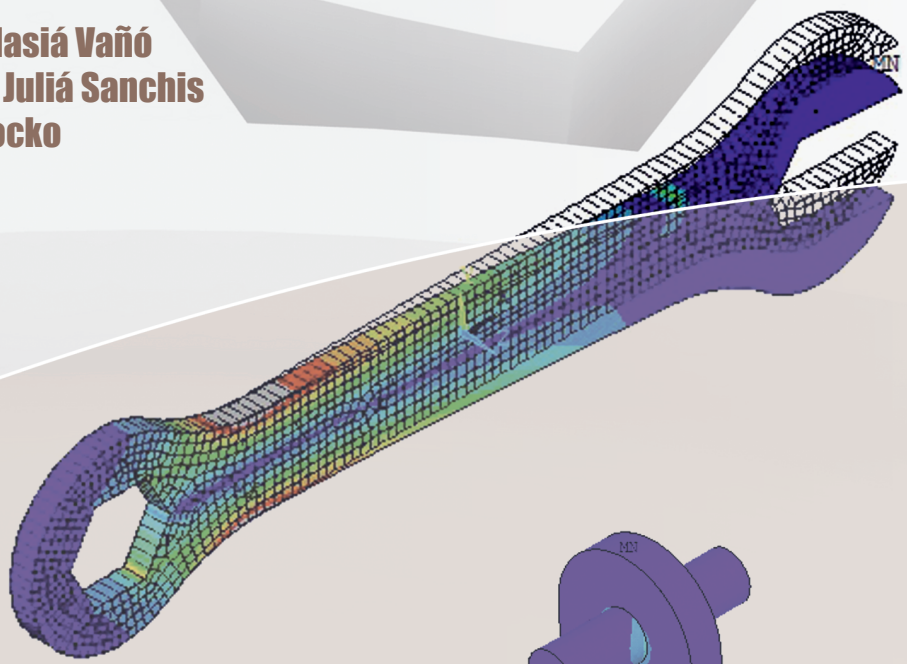
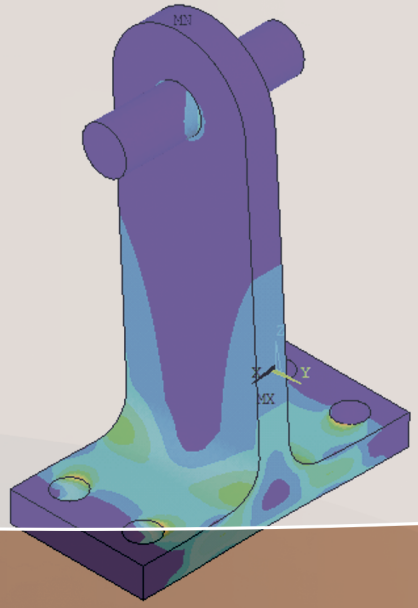
Mechanical behaviour of materials: simulation problems

Jaime Masiá Vañó
Ernesto Juliá Sanchis
Josef Bocko





EDITORIAL
UNIVERSITAT POLITÈCNICA DE VALÈNCIA

Presentation
Credits
Index
Introduction
Problems
References

PRESENTATION

In the field of Mechanical Engineering and Product Design, it is very important to optimize the models and materials, which means a reduction in the production cost.

To achieve this optimization from the structural point of view, traditionally there have been theoretical methods that have allowed a simulation of reality. One of the most significant methods is the Finite Element Method. For a practical use of this particular method, computers with enough computational power are required to solve the problems. One of the software packages that can perform this type of analysis is ANSYS. The set of exercises that are presented in this publication allow students gradually learn how to use this program. No previous experience is required.

The proposed exercises range from simple linear models to more complex solid models. They also use different types of structural elements and materials, so that the student is able to investigate their behaviour and find out which is the most appropriate solution for every application.

Based on different practical applications, the exercises of this publication pretend to be a very useful learning tool, both for students and for researchers who are working in mechanical applications of the Finite Element Method. The exercises are accompanied with lots of explanations and figures of the menus and dialog boxes. Each particular exercise provides detailed information for student to improve in the use of the software. Students will be able to think about how the program works and how to use ANSYS to solve any problem.

Some exercises consider some simplifying assumptions in order to approach the model to the real situation, and sometimes that depends on the limitations of the program and of the method.

To sum up, the aim of this electronic publication is to offer students one of the most useful tools in the field of Structural Engineering.

© Jaime Masiá Vañó, Ernesto Juliá Sanchis, Jozef Bocko

Presentation
Credits
Index
Introduction
Problems
References

CREDITS

The contents of this publication have been subject to revision by the Department of Mechanical and materials engineering Universitat Politècnica de València (Spain).

Academic book collection

To cite this publication please use the following reference: MASIÁ VAÑÓ, JAIME, [et al] (2013) Mechanical behaviour of materials: simulation problems. Valencia: Universitat Politècnica

First edition, 2013 (electronic)

© Jaime Masiá Vañó
Ernesto Juliá Sanchis
Josef Bocko

© of this edition: Editorial Universitat Politècnica de València
distribution: Tel. 963877012 / http://www.lalibreria.upv.es / Ref.: 6107-01-01-01

ISBN: 978-84-9048-148-6 (electronic)

Any unauthorized copying, distribution, marketing, editing, and in general any other exploitation, for whatever reason, of this piece of work or any part thereof, is strictly prohibited without the authors' expressed and written permission.

INDEX

Introduction

BAR ELEMENTS

Circular post
Three pinned bars
Planar truss
Space frame truss

BEAM ELEMENTS

Cantilever beam
Beam simply supported
Fink truss
Container

PIPE ELEMENTS

Swing
Chair

PLANE ELEMENTS

Plate with a hole
Custom sections
Plastic bracket
Tensile test specimen
Wrench

SHELL ELEMENTS

Triangular support bracket
Tin
Pressure tank

SOLID ELEMENTS

Snap fit
Shaft support
Pulley
Wall hanger

© Jaime Masiá Vañó, Ernesto Juliá Sanchis, Jozef Bocko

Mechanical Behaviour of Materials
Simulation Problems

INTRODUCTION TO NUMERICAL METHODS

Direct Deformation Method

Direct deformation method is predecessor of the finite element method. The method is based on determination of deformation and stresses in the force acting stiffness matrix. Structural members can be built of one or more elements, which are connected in nodes. Deformation of structure is described by

Figure 1. Loaded structure

The displacements of points on which are forces of separate structural members and they can be written in form

$$\{u\} = \{u_1, u_2, \ldots, u_n\}^T \tag{1.1}$$

where u_i $(i = 1, \ldots, n)$

For the case of linear dependency between deformations and loads there exists one or more relations

$$\{F\} = [K]\{u\} \tag{1.2}$$

Relation between forces and displacements of separate nodes can be expressed in form

$$\begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & \cdots & k_{1n} \\ k_{21} & k_{22} & \cdots & k_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ k_{n1} & k_{n2} & \cdots & k_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{Bmatrix} \tag{1.3}$$

where

$$k_{ij} = k_{ji} \tag{1.4}$$

Since component F_i of force $\{F\}$ can be represented by linear combination of components u_j of vector u and coefficients k_{ij} of matrix $[K]$ and because of symmetry of matrix $[K]$ it is possible to write equations for vector $\{u\}$ in vector form and in form of system of linear equations as it is represented by relation

$$\{u_1, u_2, \ldots, u_n\}^T = [K]^{-1}\{F\} \tag{1.5}$$

and for our linear form $\{F\} = \{1, \ldots, 1\}^T$ we obtain

$$\{u_1, u_2, \ldots, u_n\}^T = [K]^{-1}\{1, \ldots, 1\}^T \tag{1.6}$$

where numbers

$$k_{ij} = k_{ji} \tag{1.7}$$

are coefficients of linear form F with respect to the displacement u_i. The coefficients k_{ij} are coefficients of nodes that linear form F, respect to individual beam section. On the basis of Maxwell relations, we can write

$$k_{ij} = k_{ji} = \frac{\partial F_i}{\partial u_j} = \frac{\partial F_j}{\partial u_i} \tag{1.8}$$

from which results that k_{ij} are coefficients of nodes u_i corresponding to unit displacements u_j.

Isolated spring system

We consider spring system according to Figure 2 with loading forces and displacements of nodal points.

Figure 2. Spring system

For spring no. 1 figure 2 we have equations

$$F_1 = k_1 u_1 \tag{1.9}$$

or in matrix form

$$\begin{Bmatrix} F_1 \\ 0 \end{Bmatrix} = \begin{bmatrix} k_1 & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ 0 \end{Bmatrix} \tag{1.10}$$

Relations for forces in spring number 2 in form of equations and in matrix form are given by following relations

$$F_2 = k_2 (u_2 - u_1) \tag{1.11}$$

The system of equations can be written in matrix form

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k_1 & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ 0 \end{Bmatrix} + \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{1.12}$$

Stiffness matrix K in this case is the sum of the first two matrices in linear combination of linear stiffness matrices of individual springs. Stiffness matrix of the system is symmetrical and it is possible to write it in form of linear combination of matrices of individual springs. Stiffness matrix of individual springs is symmetrical and it is possible to write it in form of linear combination of matrices of individual springs.

The system is denoted two points.

For given forces F_1 and arbitrarily F_2, for we consider displacements u_1 and u_2 from equation 1.12 we can write the system of equations in form of matrix form

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k_1 & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ 0 \end{Bmatrix} + \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{1.13}$$

Generally, the system can be denoted

$$\{F\} = [K]\{u\} \tag{1.14}$$

or

$$\{u\} = [K]^{-1}\{F\} \tag{1.15}$$

Because we want to know displacements u_i, from the force equation 1.15 we have

$$\{u\} = [K]^{-1}\{F\} \tag{1.16}$$

Inverse matrix equation can be used for computation of unknown forces. From the second equation we have

$$\{F\} = [K]\{u\} \tag{1.17}$$

If rank $K_1 = 2$, equation 1.17 can be written in form

$$\{F\} = [K]^{-1}\{u\} \tag{1.18}$$

Device movement of structure is determined by following steps:

1. Number of nodes and number of unknown displacements
2. Determination of stiffness matrix of individual beam sections and determination of forces and displacements in individual nodes
3. Determination of stiffness matrix of individual beam sections
4. Assembling of stiffness matrix and establishing of global system of equations
5. Determination of unknown displacements
6. Solution of equation system for unknowns u_i and F_i

Finite Element Method

The Finite Element Method (FEM) is the most spread method in structural mechanics. It is the product of the 20th century. The first publication is from 1943 by Hrennikoff and Courant 1943 and 1950 and Turner et al. 1956.

Figure 3. Division of structure into separate elements

The method which directly is based on the use of stiffness matrices. The basic idea of the method is to divide the structure into separate elements, which are connected in nodes. Deformation of structure is described by the system of equations

$$[K]\{u\} = \{F\} \tag{2.1}$$

where K is a matrix of stiffness and F is a vector of forces.

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} & \cdots & k_{1n} \\ k_{21} & k_{22} & k_{23} & \cdots & k_{2n} \\ k_{31} & k_{32} & k_{33} & \cdots & k_{3n} \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ k_{n1} & k_{n2} & k_{n3} & \cdots & k_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ \vdots \\ F_n \end{Bmatrix} \tag{2.2}$$

or in short form

$$[K]\{u\} = \{F\} \tag{2.3}$$

where K is the stiffness matrix, u is a column of displacement

$$\{u\} = \{u_1, u_2, u_3, \ldots, u_n\}^T \tag{2.4}$$

and F is a column of forces

$$\{F\} = \{F_1, F_2, F_3, \ldots, F_n\}^T \tag{2.5}$$

In order of stiffness forces acting on nodes u_i is a column of surface, k a column of stiffness. If k_{ij} is a column of stiffness matrix K and u_j is a column of displacement, then $k_{ij} u_j$ is a column of forces acting on i th column of nodal points of system. Single functions u_i and F_i are functions of time and they can be dependent on other variables. Stiffness matrix K is symmetrical and it is possible to write it in form of linear combination of matrices of individual springs.

Stiffness matrix K is a matrix of single functions and it is a column of displacement of nodal points. Because the first equation of system 2.2 is

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} & \cdots & k_{1n} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} F_1 \end{Bmatrix} \tag{2.6}$$

we can write relation

$$\{k_{11} u_1 + k_{12} u_2 + k_{13} u_3 + \cdots + k_{1n} u_n\} = \{F_1\} \tag{2.7}$$

which represents dependence of force F_1 on all nodal points in direction of nodal points u.

Matrix K means all dependence of forces functions. In the main we suppose linear models, without dependence on time.

Matrix K is a matrix of elastic constants.

Matrix K is a matrix of stiffness.

Application of finite element 2D problem

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.8}$$

Because matrix K is dependent on the type of deformation and on the type of material, it can be defined by relation including previous relation with the type of material

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.9}$$

where K is a matrix of stiffness forces acting structure, ε with prescribed tensions. Variations $\delta\varepsilon$ and δu are connected with relation which is valid for K matrix

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.10}$$

where ε is element stiffness matrix and δ is column of loading in nodal points. Stress matrix can be written in form of product of matrix D and matrix B and displacement u

$$\{\sigma\} = [\mathbf{D}][\mathbf{B}]\{u\} \tag{2.11}$$

and forces

$$\{\varepsilon\} = [\mathbf{B}]\{u\} \tag{2.12}$$

For the whole body we have equation

$$\{\delta\} = [\mathbf{K}]\{u\} \tag{2.13}$$

where $[\mathbf{K}] = \int_V [\mathbf{B}]^T [\mathbf{D}][\mathbf{B}]\, dV$ is a matrix of stiffness acting in nodal forces and u is vector corresponding displacement of nodal points. Stiffness matrix K is symmetrical and it is possible to write it in form of linear combination of matrices of individual springs. Stiffness matrix K is a matrix of constants in the form so which global constants system. Similar case can be dealed for equations of equilibrium and for equations of compatibility.

Stiffness matrix of four-noded beam

Let us compute general stiffness matrix of two element in plane, $x = \{x, y\}^T$.

Figure 4. Four element

At the beginning we suppose that the displacements of point point of element is direction i can be expressed in form of polynomial

$$u_i = a_1 + a_2 x_i + a_3 y_i \quad \text{where} \quad x_i = x_1, x_2, x_3, x_4 \tag{2.14}$$

where a_1, a_2 and a_3 are displacements of point i and x and y components i, length of one side. Nodal point i and k have coordinates x_i and y_i, respectively. The length of one side is then $x_k - x_i$ and $y_k - y_i$.

and direction cosines of element are

$$l_{ik} = \frac{x_k - x_i}{L_{ik}} \qquad m_{ik} = \frac{y_k - y_i}{L_{ik}} \tag{2.15}$$

where L_{ik} is length of element ik.

Displacements of nodal points in direction i can be, on the basis of figure 4, expressed by equations

$$u_1 = a_1 + a_2 x_1 + a_3 y_1$$
$$u_2 = a_1 + a_2 x_2 + a_3 y_2$$
$$u_3 = a_1 + a_2 x_3 + a_3 y_3$$
$$u_4 = a_1 + a_2 x_4 + a_3 y_4 \tag{2.16}$$

where u_1, u_2, u_3, u_4 are displacements of nodes x_1, x_2, x_3, x_4 and y_1, y_2, y_3, y_4 are coordinates of nodal points. Relation between displacements and coordinates is

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} \tag{2.17}$$

Using inverse of matrix

$$\begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix}^{-1} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} \tag{2.18}$$

Matrix $\mathbf{B}$ transforms displacements of nodal points into stress according to relation

$$\{\varepsilon\} = [\mathbf{B}]\{u\} \tag{2.19}$$

After substitution of equation 2.18 into 2.19 we have

$$\{\varepsilon\} = [\mathbf{B}]\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix}^{-1} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} \tag{2.20}$$

Matrix $\mathbf{B}$ is

$$[\mathbf{B}] = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 0 & 1 \\ 0 & 1 & 1 \end{bmatrix} \tag{2.21}$$

where ε is column of strain $\varepsilon = \{\varepsilon_x, \varepsilon_y, \gamma_{xy}\}$ and u is column of displacement of nodal points. Because the first equation of system 2.16 is

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} \tag{2.22}$$

we can write relation

$$\{u_1 = a_1 + a_2 x_1 + a_3 y_1\} \tag{2.23}$$

which represents dependence of stress in direction x on displacements of nodal points u.

Matrix $\mathbf{B}$ means all dependence of stress functions. In the main we suppose linear models, without dependence on time.

Matrix $\mathbf{B}$ is a matrix of elastic constants.

Matrix $\mathbf{B}$ is a matrix of stiffness.

Application of finite element 2D problem

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.24}$$

Because matrix $\mathbf{B}$ is dependent on the type of deformation and on the type of material, it can be defined by relation including previous relation with the type of material

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.25}$$

where K is a matrix of stiffness forces acting structure, ε with prescribed tensions. Variations $\delta\varepsilon$ and δu are connected with relation which is valid for K matrix

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.26}$$

where ε is element stiffness matrix and δ is column of loading in nodal points. Stress matrix can be written in form of product of matrix D and matrix B and displacement u

$$\{\sigma\} = [\mathbf{D}][\mathbf{B}]\{u\} \tag{2.27}$$

and forces

$$\{\varepsilon\} = [\mathbf{B}]\{u\} \tag{2.28}$$

For the whole body we have equation

$$\{\delta\} = [\mathbf{K}]\{u\} \tag{2.29}$$

where $[\mathbf{K}] = \int_V [\mathbf{B}]^T [\mathbf{D}][\mathbf{B}]\, dV$ is a matrix of stiffness acting in nodal forces and u is vector corresponding displacement of nodal points. Stiffness matrix K is symmetrical and it is possible to write it in form of linear combination of matrices of individual springs. Stiffness matrix K is a matrix of constants in the form so which global constants system. Similar case can be dealed for equations of equilibrium and for equations of compatibility.

Stiffness matrix of four-noded beam

Let us compute general stiffness matrix of two element in plane, $x = \{x, y\}^T$.

Figure 5. Four element

At the beginning we suppose that the displacements of point point of element is direction i can be expressed in form of polynomial

$$u_i = a_1 + a_2 x_i + a_3 y_i \quad \text{where} \quad x_i = x_1, x_2, x_3, x_4 \tag{2.30}$$

where a_1, a_2 and a_3 are displacements of point i and x components i, length of one side. Nodal point i and k have coordinates x_i and y_i, respectively. The length of one side is then $x_k - x_i$ and $y_k - y_i$.

and direction cosines of element are

$$l_{ik} = \frac{x_k - x_i}{L_{ik}} \qquad m_{ik} = \frac{y_k - y_i}{L_{ik}} \tag{2.31}$$

where L_{ik} is length of element ik.

Displacements of nodal points in direction i can be, on the basis of figure 5, expressed by equations

$$u_1 = a_1 + a_2 x_1 + a_3 y_1$$
$$u_2 = a_1 + a_2 x_2 + a_3 y_2$$
$$u_3 = a_1 + a_2 x_3 + a_3 y_3$$
$$u_4 = a_1 + a_2 x_4 + a_3 y_4 \tag{2.32}$$

where u_1, u_2, u_3, u_4 are displacements of nodes x_1, x_2, x_3, x_4 and y_1, y_2, y_3, y_4 are coordinates of nodal points. Relation between displacements and coordinates is

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} \tag{2.33}$$

Using inverse of matrix

$$\begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix}^{-1} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} \tag{2.34}$$

Matrix $\mathbf{B}$ transforms displacements of nodal points into stress according to relation

$$\{\varepsilon\} = [\mathbf{B}]\{u\} \tag{2.35}$$

After substitution of equation 2.34 into 2.35 we have

$$\{\varepsilon\} = [\mathbf{B}]\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix}^{-1} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} \tag{2.36}$$

Matrix $\mathbf{B}$ is

$$[\mathbf{B}] = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 0 & 1 \\ 0 & 1 & 1 \end{bmatrix} \tag{2.37}$$

where ε is column of strain $\varepsilon = \{\varepsilon_x, \varepsilon_y, \gamma_{xy}\}$ and u is column of displacement of nodal points. Because the first equation of system 2.32 is

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \\ 1 & x_4 & y_4 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} \tag{2.38}$$

we can write relation

$$\{u_1 = a_1 + a_2 x_1 + a_3 y_1\} \tag{2.39}$$

which represents dependence of stress in direction x on displacements of nodal points u.

Matrix $\mathbf{B}$ means all dependence of stress functions. In the main we suppose linear models, without dependence on time.

Matrix $\mathbf{B}$ is a matrix of elastic constants.

Matrix $\mathbf{B}$ is a matrix of stiffness.

Application of finite element 2D problem

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.40}$$

Because matrix $\mathbf{B}$ is dependent on the type of deformation and on the type of material, it can be defined by relation including previous relation with the type of material

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.41}$$

where K is a matrix of stiffness forces acting structure, ε with prescribed tensions. Variations $\delta\varepsilon$ and δu are connected with relation which is valid for K matrix

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.42}$$

where ε is element stiffness matrix and δ is column of loading in nodal points. Stress matrix can be written in form of product of matrix D and matrix B and displacement u

$$\{\sigma\} = [\mathbf{D}][\mathbf{B}]\{u\} \tag{2.43}$$

and forces

$$\{\varepsilon\} = [\mathbf{B}]\{u\} \tag{2.44}$$

For the whole body we have equation

$$\{\delta\} = [\mathbf{K}]\{u\} \tag{2.45}$$

where $[\mathbf{K}] = \int_V [\mathbf{B}]^T [\mathbf{D}][\mathbf{B}]\, dV$ is a matrix of stiffness acting in nodal forces and u is vector corresponding displacement of nodal points. Stiffness matrix K is symmetrical and it is possible to write it in form of linear combination of matrices of individual springs. Stiffness matrix K is a matrix of constants in the form so which global constants system. Similar case can be dealed for equations of equilibrium and for equations of compatibility.

Stiffness matrix of triangular element

Let us compute general stiffness matrix of triangular element in plane, $x = \{x, y\}^T$.

Figure 6. Triangular element

At the beginning we suppose that the displacements of point point of element is direction i can be expressed in form of polynomial

$$u_i = a_1 + a_2 x_i + a_3 y_i \quad \text{where} \quad x_i = x_1, x_2, x_3 \tag{2.46}$$

where a_1, a_2 and a_3 are displacements of point i and x and y components i. Nodal point i and k have coordinates x_i and y_i, respectively. The length of one side is then $x_k - x_i$ and $y_k - y_i$.

and direction cosines of element are

$$l_{ik} = \frac{x_k - x_i}{L_{ik}} \qquad m_{ik} = \frac{y_k - y_i}{L_{ik}} \tag{2.47}$$

where L_{ik} is length of element ik.

Displacements of nodal points in direction i can be, on the basis of figure 6, expressed by equations

$$u_1 = a_1 + a_2 x_1 + a_3 y_1$$
$$u_2 = a_1 + a_2 x_2 + a_3 y_2$$
$$u_3 = a_1 + a_2 x_3 + a_3 y_3 \tag{2.48}$$

where u_1, u_2, u_3 are displacements of nodes x_1, x_2, x_3 and y_1, y_2, y_3 are coordinates of nodal points. Relation between displacements and coordinates is

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} \tag{2.49}$$

Using inverse of matrix

$$\begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}^{-1} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{2.50}$$

Matrix $\mathbf{B}$ transforms displacements of nodal points into stress according to relation

$$\{\varepsilon\} = [\mathbf{B}]\{u\} \tag{2.51}$$

After substitution of equation 2.50 into 2.51 we have

$$\{\varepsilon\} = [\mathbf{B}]\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}^{-1} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{2.52}$$

Matrix $\mathbf{B}$ is

$$[\mathbf{B}] = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 0 & 1 \\ 0 & 1 & 1 \end{bmatrix} \tag{2.53}$$

where ε is column of strain $\varepsilon = \{\varepsilon_x, \varepsilon_y, \gamma_{xy}\}$ and u is column of displacement of nodal points. Because the first equation of system 2.48 is

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix} \tag{2.54}$$

we can write relation

$$\{u_1 = a_1 + a_2 x_1 + a_3 y_1\} \tag{2.55}$$

which represents dependence of stress in direction x on displacements of nodal points u.

Matrix $\mathbf{B}$ means all dependence of stress functions. In the main we suppose linear models, without dependence on time.

Matrix $\mathbf{B}$ is a matrix of elastic constants.

Matrix $\mathbf{B}$ is a matrix of stiffness.

Application of finite element 2D problem

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.56}$$

Because matrix $\mathbf{B}$ is dependent on the type of deformation and on the type of material, it can be defined by relation including previous relation with the type of material

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.57}$$

where K is a matrix of stiffness forces acting structure, ε with prescribed tensions. Variations $\delta\varepsilon$ and δu are connected with relation which is valid for K matrix

$$\{\sigma\} = [\mathbf{D}]\{\varepsilon\} = [\mathbf{D}][\mathbf{B}]\{u\} = [\mathbf{K}]\{u\} \tag{2.58}$$

where ε is element stiffness matrix and δ is column of loading in nodal points. Stress matrix can be written in form of product of matrix D and matrix B and displacement u

$$\{\sigma\} = [\mathbf{D}][\mathbf{B}]\{u\} \tag{2.59}$$

and forces

$$\{\varepsilon\} = [\mathbf{B}]\{u\} \tag{2.60}$$

For the whole body we have equation

$$\{\delta\} = [\mathbf{K}]\{u\} \tag{2.61}$$

where $[\mathbf{K}] = \int_V [\mathbf{B}]^T [\mathbf{D}][\mathbf{B}]\, dV$ is a matrix of stiffness acting in nodal forces and u is vector corresponding displacement of nodal points. Stiffness matrix K is symmetrical and it is possible to write it in form of linear combination of matrices of individual springs. Stiffness matrix K is a matrix of constants in the form so which global constants system. Similar case can be dealed for equations of equilibrium and for equations of compatibility.

Stiffness matrix of triangular element

According to integration form

- analytic (only special elements),
- numerical (general).

There are very different elements based on the basis of polynomial, trigonometric, hyperbolic, Bessel functions, Lagrange, Hermite, Lagrange-Hermite, Gauss-Legendre, Gauss-Lobatto, Gauss-Jacobi, Gauss-Chebyshev, Newton-Cotes, Chebyshev, Lobatto, Jacobi, Bessel, Fourier, Chebyshev, Legendre, Laguerre, Hermite, Lagrange

Name: Ashish Kumar Sahoo
Roll No: 18ME10
Mobile: 8895-
Email: 18me10@
Branch: Mechanical Engineering

CIRCULAR PLATE

Figure 1: Circular plate with Dimensions

Figure 2 is a circular plate made of Mild Steel with different holes. Use 1.5 dimensional sketch. Use 3-D Solid Extrude to create the plate.

Figure 2: Circular plate

Figure 3 is a plot of the mechanical properties of Mild Steel. Use the properties corresponding to Mild Steel.

Table 1: Mild Steel Properties

Mechanical Properties		Physical Properties	
Young's Modulus	200 GPa	Density	7850 kg/m³
Poisson's Ratio	0.29	Mass Density	7850 kg/m³
Shear Modulus	77.5 GPa	Thermal Conductivity	46.6 W/m·K
Tensile Strength	250 MPa	Specific Heat	434 J/kg·K
Yield Strength	250 MPa	Thermal Expansion Coefficient	11.7 µm/m·K

Table 2: Mild Steel Properties

Find out maximum Load of Mild Steel. Calculate:

1. Maximum Load Capacity
2. Factor of safety
3. Deformation
4. Stress Distribution

ACCORDING TO THE MESH MODEL

Figure 4 is a mesh of Circular Plate. Use Mesh Type as Tetrahedral. Use Mesh Size as 0.5.

Figure 4: Circular plate Mesh

Figure 5 is a Fixture Plot. Use Fixture Type as Fixed Geometry.

Figure 5: Circular plate Fixture

Figure 6 is a Fixture Plot of Circular Plate. Use Fixture Type as Fixed Geometry.

Figure 6: Circular plate Fixture

Figure 7 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 7: Circular plate Load

Figure 8 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 8: Circular plate Load

Figure 9 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 9: Circular plate Load

Figure 10 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 10: Circular plate Load

Figure 11 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 11: Circular plate Load

Find out the Load Capacity of the Circular plate.

Load Capacity = 1000 N

Find out the deformation of the Circular plate.

Figure 12: Circular plate Deformation

Find out the Factor of safety of the Circular plate.

Figure 13: Circular plate Factor of safety

Find out the Stress Distribution of the Circular plate.

Figure 14: Circular plate Stress

Find out the Equivalent Stress of the Circular plate.

Figure 15: Circular plate Equivalent Stress

Find out the Maximum Shear Stress of the Circular plate.

Figure 16: Circular plate Maximum Shear Stress

Find out the Normal Stress of the Circular plate.

Figure 17: Circular plate Normal Stress

Find out the Shear Stress of the Circular plate.

Figure 18: Circular plate Shear Stress

ACCORDING TO THE MESH MODEL

Figure 19 is a mesh of Circular Plate. Use Mesh Type as Tetrahedral. Use Mesh Size as 0.5.

Figure 19: Circular plate Mesh

Figure 20 is a Fixture Plot of Circular Plate. Use Fixture Type as Fixed Geometry.

Figure 20: Circular plate Fixture

Figure 21 is a Fixture Plot of Circular Plate. Use Fixture Type as Fixed Geometry.

Figure 21: Circular plate Fixture

Figure 22 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 22: Circular plate Load

Figure 23 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 23: Circular plate Load

Figure 24 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 24: Circular plate Load

Figure 25 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 25: Circular plate Load

Find out the Load Capacity of the Circular plate.

Load Capacity = 1000 N

Find out the deformation of the Circular plate.

Figure 26: Circular plate Deformation

Find out the Factor of safety of the Circular plate.

Figure 27: Circular plate Factor of safety

Find out the Stress Distribution of the Circular plate.

Figure 28: Circular plate Stress

Find out the Equivalent Stress of the Circular plate.

Figure 29: Circular plate Equivalent Stress

Find out the Maximum Shear Stress of the Circular plate.

Figure 30: Circular plate Maximum Shear Stress

Find out the Normal Stress of the Circular plate.

Figure 31: Circular plate Normal Stress

Find out the Shear Stress of the Circular plate.

Figure 32: Circular plate Shear Stress

ACCORDING TO THE MESH MODEL

Figure 33 is a mesh of Circular Plate. Use Mesh Type as Tetrahedral. Use Mesh Size as 0.5.

Figure 33: Circular plate Mesh

Figure 34 is a Fixture Plot of Circular Plate. Use Fixture Type as Fixed Geometry.

Figure 34: Circular plate Fixture

Figure 35 is a Fixture Plot of Circular Plate. Use Fixture Type as Fixed Geometry.

Figure 35: Circular plate Fixture

Figure 36 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 36: Circular plate Load

Figure 37 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 37: Circular plate Load

Figure 38 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 38: Circular plate Load

Figure 39 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 39: Circular plate Load

Find out the Load Capacity of the Circular plate.

Load Capacity = 1000 N

Find out the deformation of the Circular plate.

Figure 40: Circular plate Deformation

Find out the Factor of safety of the Circular plate.

Figure 41: Circular plate Factor of safety

Find out the Stress Distribution of the Circular plate.

Figure 42: Circular plate Stress

Find out the Equivalent Stress of the Circular plate.

Figure 43: Circular plate Equivalent Stress

Find out the Maximum Shear Stress of the Circular plate.

Figure 44: Circular plate Maximum Shear Stress

Find out the Normal Stress of the Circular plate.

Figure 45: Circular plate Normal Stress

Find out the Shear Stress of the Circular plate.

Figure 46: Circular plate Shear Stress

ACCORDING TO THE MESH MODEL

Figure 47 is a mesh of Circular Plate. Use Mesh Type as Tetrahedral. Use Mesh Size as 0.5.

Figure 47: Circular plate Mesh

Figure 48 is a Fixture Plot of Circular Plate. Use Fixture Type as Fixed Geometry.

Figure 48: Circular plate Fixture

Figure 49 is a Fixture Plot of Circular Plate. Use Fixture Type as Fixed Geometry.

Figure 49: Circular plate Fixture

Figure 50 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 50: Circular plate Load

Figure 51 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 51: Circular plate Load

Figure 52 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 52: Circular plate Load

Figure 53 is a Load Plot of Circular Plate. Use Load Type as Force and Load Value as 1000 N.

Figure 53: Circular plate Load

Find out the Load Capacity of the Circular plate.

Load Capacity = 1000 N

Find out the deformation of the Circular plate.

Figure 54: Circular plate Deformation

Find out the Factor of safety of the Circular plate.

Figure 55: Circular plate Factor of safety

Find out the Stress Distribution of the Circular plate.

Figure 56: Circular plate Stress

Find out the Equivalent Stress of the Circular plate.

Figure 57: Circular plate Equivalent Stress

Find out the Maximum Shear Stress of the Circular plate.

Figure 58: Circular plate Maximum Shear Stress

Find out the Normal Stress of the Circular plate.

Figure 59: Circular plate Normal Stress

Find out the Shear Stress of the Circular plate.

Figure 60: Circular plate Shear Stress

Finally, to represent the distribution of stresses in the model, it is used the option "Plot Elem Table" (Figure 31):

Main Menu > General Postproc > Element Table > Plot Elem Table

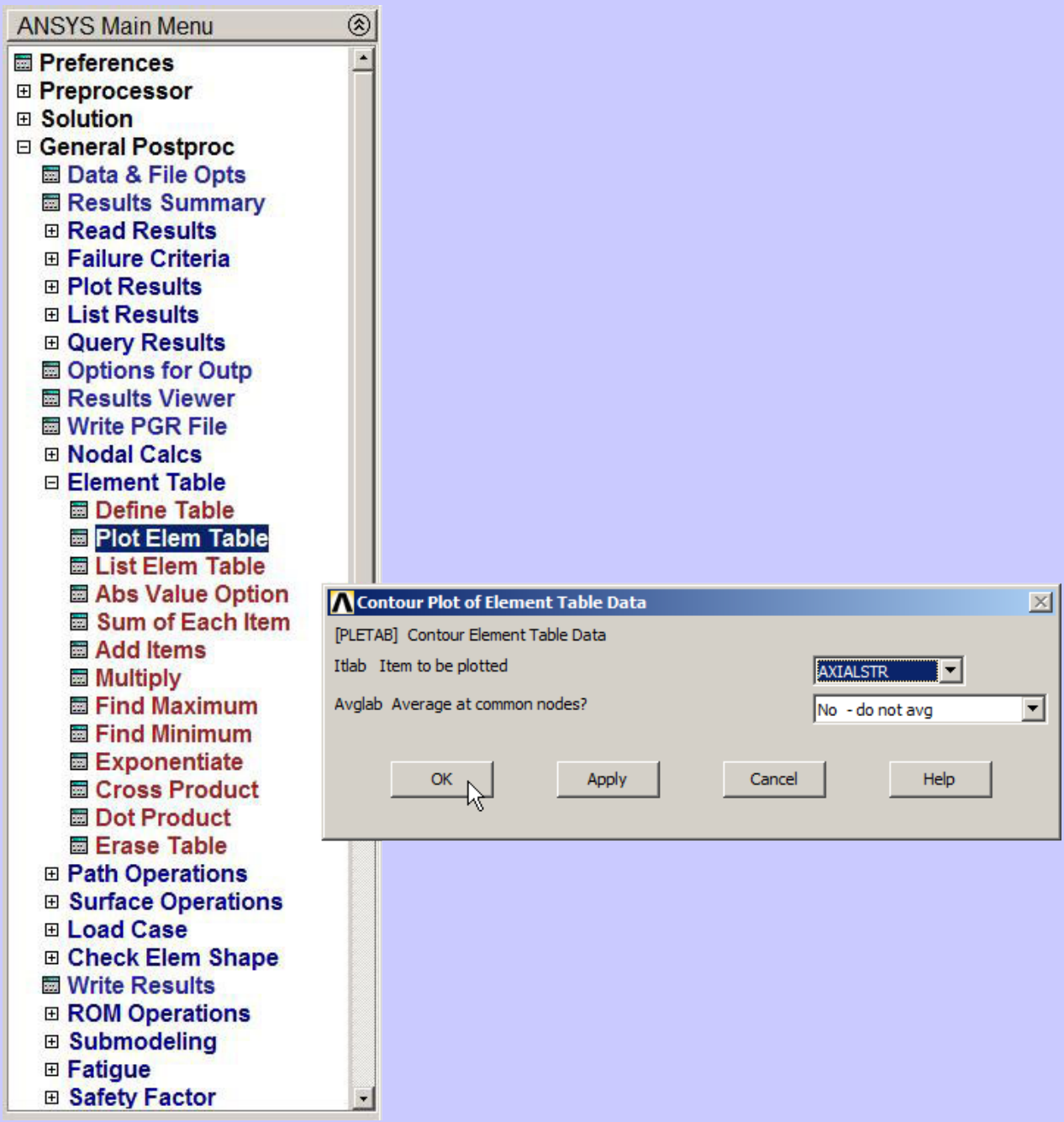


Figure 31. "Plot Element Table".

And the distribution of stresses in the circular post is showed in Figure 32.

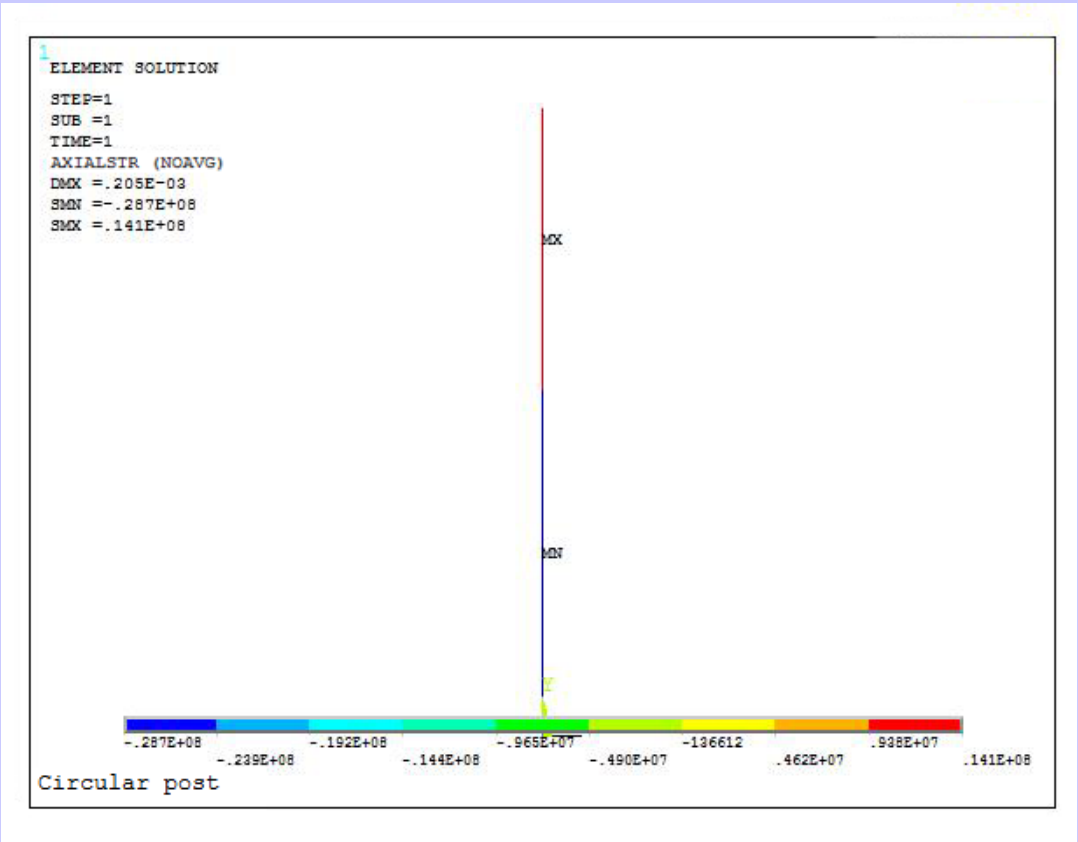


Figure 32. Stress distribution in the model.

The results indicate that the aluminium bar supports 28.7 MPa. This is the maximum value, what means that the post is safe since the elastic limit for the material is 275 MPa.

© Jaime Masiá Vañó, Ernesto Juliá Sanchis, Jozef Bocko

Para seguir leyendo haga click aquí