

USING OF FINITE ELEMENT METHOD FOR SOLVING PROBLEMS OF THE RESISTANCE OF MATERIALS.

Pronkevich S.A., Belarusian National Technical University, Minsk;
Senk I, Faculty of Technical Science, Novi Sad.

In this article we consider the using of finite element method for solving problems of the resistance of materials and compare results solved different methods: analytic solution, hand-calculation solution by finite element method and solution in ANSYS.

The wide distribution of modern computer technic has changed the engineering appreciably. The using of modern computer methods provides capability to research different characteristics of projected objects. It allows changing construction of this objects without prototyping and long and expensive simulations.

The finite element method takes up important place among modern computing methods. The finite element method is a numerical procedure that can be used to obtain solution to a wide class of engineering problems involving stress analysis, heat transfer, fluid flow, etc.

Consider an example of solution of a simple problem of the resistance of materials by finite element method with hand-calculation, with ANSYS and compare the results with analytical solution.

Consider a bar with a variable cross section supporting a load P , as shown in Figure 1. The bar is fixed at one end and carries the load P at the other end. Let us designate the width of the bar at the top by w_1 at the bottom by the w_2 , its thickness by t , and its length by L . The bar's modulus of elasticity will be denoted by E . We are interested in determining how much the bar will deflect at various points along its length when it is subjected to the load P . We will neglect the weight of the bar in the following analysis, assuming that the applied load is considerably larger than the weight of the bar.

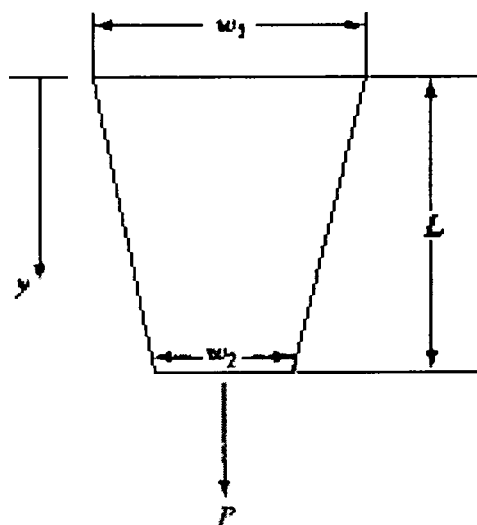


Figure 1. A bar under axial loading

The given bar is modeled using four individual segments, with each segment having a uniform cross section. The cross-sectional area of each element is represented by an average area of the cross section at the nodes that make up (define) the element. This model is shown in Figure 2.

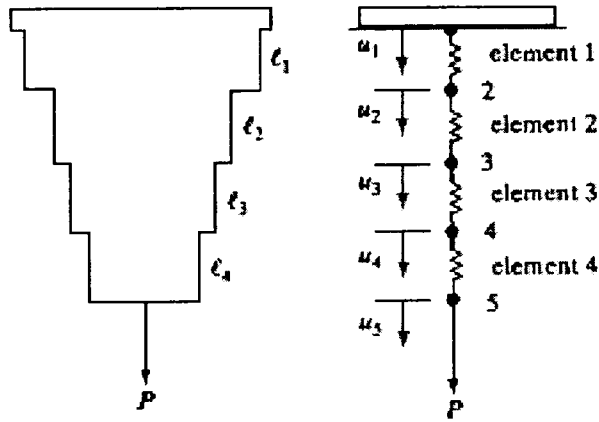


Figure 2. Subdividing the bar into elements and nodes

In order to obtain numerical values of the nodal displacement, let us assume that $E = 10.4 \cdot 10^6 \frac{lb}{in^2}$, $w_1 = 2 in$, $w_2 = 1 in$, $t = 0.125 in$, $L = 10 in$ ($l = 2.5 in$), $P = 1000 lb$

The variation of the cross-section area of the bar in the y – direction can be expressed by

$$A(y) = (w_1 + (\frac{w_2 - w_1}{L})y)t = (2 + \frac{(1-2)}{10}y) \cdot 0.125 = 0.25 - 0.0125y \quad (*)$$

Using (*) we can compute the cross-section areas at each node:

$$A_1 = 0.25 in^2 \quad A_2 = 0.21875 in^2$$

$$A_3 = 0.1875 in^2 \quad A_4 = 0.15625 in^2$$

$$A_5 = 0.125 in^2$$

Next, the equivalent stiffness coefficient for each element is computed from the equations

$$k_i = \frac{A_{i+1} + A_i}{2l} E$$

$$k_1 = 975 \cdot 10^3 \frac{lb}{in} \quad k_2 = 845 \cdot 10^3 \frac{lb}{in}$$

$$k_3 = 715 \cdot 10^3 \frac{lb}{in} \quad k_4 = 585 \cdot 10^3 \frac{lb}{in}$$

The global stiffness matrix is obtained by assembling, or adding, together each element's position in the global stiffness matrix:

$$[K]^{(G)} = [K]^{(1G)} + [K]^{(2G)} + [K]^{(3G)} + [K]^{(4G)}$$

$$[K]^{(G)} = \begin{bmatrix} k_1 & -k_1 & 0 & 0 & 0 \\ -k_1 & k_1 + k_2 & -k_2 & 0 & 0 \\ 0 & -k_2 & k_2 + k_3 & -k_3 & 0 \\ 0 & 0 & -k_3 & k_3 + k_4 & -k_4 \\ 0 & 0 & 0 & -k_4 & k_4 \end{bmatrix}$$

Applying the boundary conditions $u_1 = 0$ and the load $P = 1000 lb$ we get

$$10^3 \begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ -975 & 1820 & -845 & 0 & 0 \\ 0 & -845 & 1560 & -715 & 0 \\ 0 & 0 & -715 & 1300 & -585 \\ 0 & 0 & 0 & -585 & 585 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 10^3 \end{Bmatrix}$$

If we will have solved this matrix we find displacement:
 $u_1 = 0$, $u_2 = 0.001026in$, $u_3 = 0.002210in$, $u_4 = 0.003608in$, $u_5 = 0.005317in$

Modeling in ANSYS consists of the following steps:

1. to select the type of finite element – “link1”.
2. to set material properties. In this task we use elastic isotropic material and input modulus of elasticity and Poisson's ratio.
3. because the model consists of 4 finite elements with different cross-sections we have to set up an individual set of real constants for each element and apply it while creating the elements.
4. According to problem situation we fix rigidly the upper node and apply load P to the lower node.

Model is shown on Figure 3.

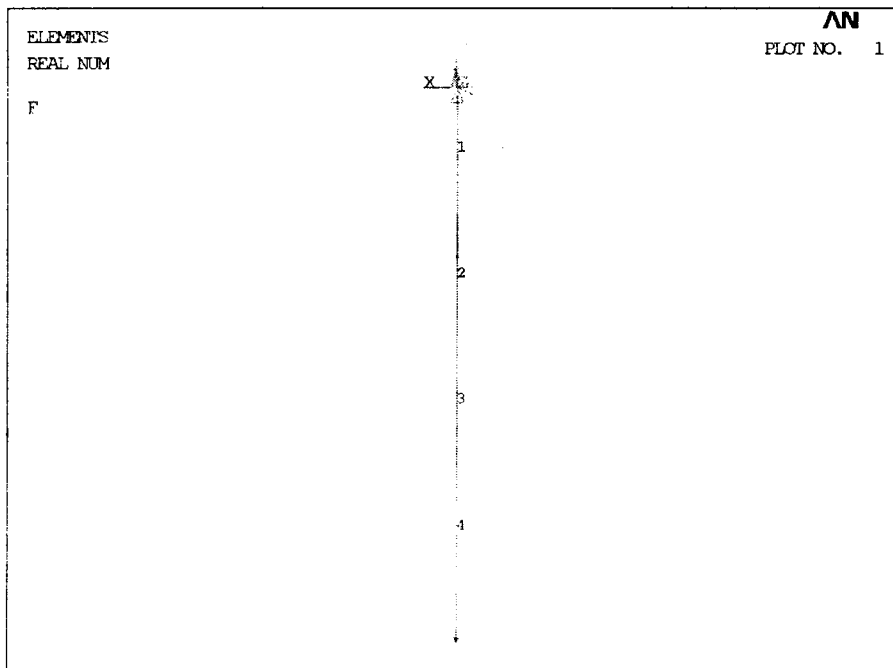


Figure 3. Finite elements model created in ANSYS.

The results of the nodes displacement are shown on Figure 4.

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

NODE	UY
1	0.0000
2	-0.10256E-02
3	-0.22091E-02
4	-0.36077E-02
5	-0.53171E-02

Figure 4. The results of the nodes displacement.

Consider an analytic solution of this problem. Assuming that the material of the bar meets Hook's law we can use the following formulae to the calculation of bar displacement $\Delta l = \int \frac{Pdy}{EA(y)}$ or in

our model for each step $\Delta l_i = \int_0^{2.5i} \frac{1000}{10.4 \cdot 10^6 \cdot (0.25 - 0.0125y)} dy$

Calculating for each step we will find:

- $\Delta l_1 = 0$ in
- $\Delta l_2 = 0.00102716$ in
- $\Delta l_3 = 0.00221294$ in
- $\Delta l_4 = 0.00361541$ in
- $\Delta l_5 = 0.0053319$ in

Compare results:

	1	2	3	4	5
FEA	0	0.001026	0.002210	0.003608	0.005317
ANSYS	0	0.0010256	0.0022091	0.0036077	0.0053171
Analytic	0	0.00102761	0.00221294	0.00361541	0.0053319

We can see the solution results of different methods are equal.

References

Миролюбов И.Н. и др. Пособие к решению задач по сопротивлению материалов. «Высшая школа» 1967 г.
 Saeed Moaveni. Finite Element Analysis. Theory and Application with ANSYS. Minnesota State University, Mancato.
 ANSYS User's Manual.