

Abstract

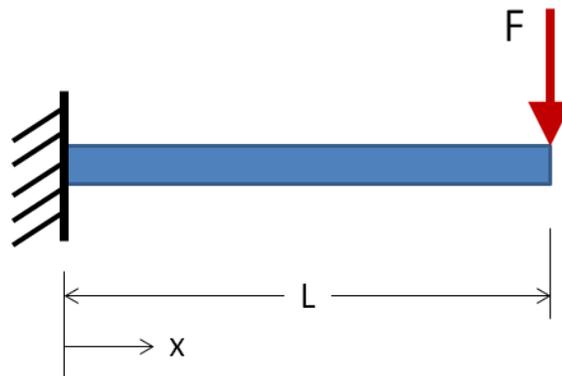
The Finite Element Method (FEM) - the main method of modern structural mechanics underlying the vast majority of modern software systems designed to perform the calculations of building structures on the computer.

But the range of its application is very wide: the construction and mechanical engineering, hydro and aerodynamics, mining and the latest technology, as well as various problems of mathematical physics - thermal conductivity, filtration, propagation, etc...

The finite element method was first applied in engineering practice in the early 50-ies. XX century. It was originally developed by two independent areas - engineering and mathematics. In the early stages the FEM formulation based on the principles of structural mechanics, limiting its scope. Only when the foundations have been formulated in the variational method of the form, it became possible to spread it to many other tasks. The rapid development of MCE paralleled the progress of modern computer technology and its application in various fields of science and engineering practice.

Problem statement

For our problem we will analyze the cantilever beam with the applied force at the edge.



Input data:

Material: Aluminum

$E = 69 \cdot 10^3$ (Modulus of Elasticity, MPa)

$D = 10$ (Diameter, mm)

$P = 100$ (Load, N)

$L = 150$ (Length, mm)

$x =$ Some distance as indicated, mm

Theoretical solution

Maximal deflection at the unsupported end:

$$y_{\max} = \frac{Pl^3}{3EI} = \frac{100 \cdot 150^3 \cdot 64}{3 \cdot 69 \cdot 10^3 \cdot \pi \cdot 10^4} = 3,32 \text{ (mm)}$$

Maximal Stress at x=0:

$$s_{\max} = \frac{Pl}{W} = \frac{100 \cdot 150 \cdot 32}{\pi \cdot 10^3} = 152,8 \text{ (MPa)}$$

Matlab solution

```
clc
clear
E=69000;           % Young's modulus, MPa
D=10;             % Diameter of the beam, mm
A=0.25*(pi)*(D)^2; % Cross area of the beam, mm^2
I=(pi/64)*(D)^4;  % Inertial moment of the beam, mm^4
L=150;           % Length of the beam, mm
P=-100;          % Applied force, N
x1=150;          % Distance for deflection calculations, mm
x2=0;            % Distance for stress calculations, mm
PP=(P/(6*E*I));
M=100;
DX=L/M;
for i=1:M;
    X(i)=i*DX;
    Y(i)=PP*((X(i))^2)*(3*L-X(i))
end
deflection=PP*((x1)^2)*(3*L-x1)
stress=P*(L-x2)*(D/2)/I % Only maximal stress in cross section
calculated
plot(X,Y)
grid on
xlabel('X Distance (mm)')
ylabel('Deflection (mm)')
axis equal
```

Result:

deflection =

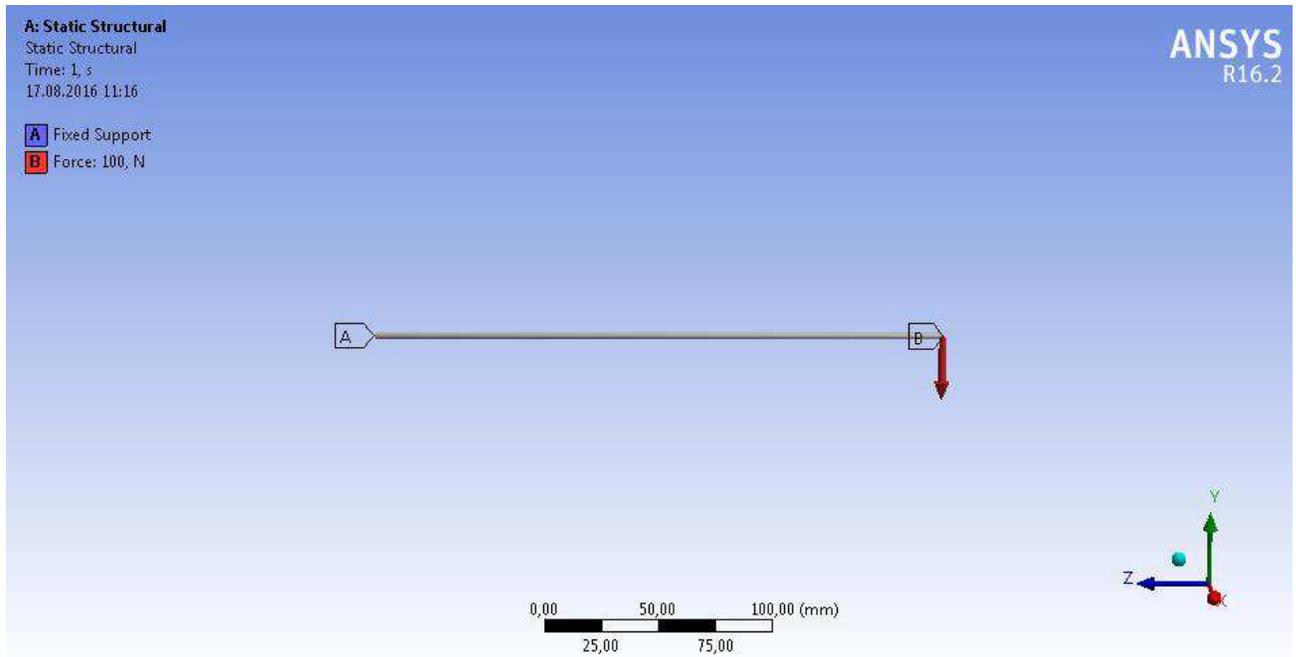
-3.3215

stress =

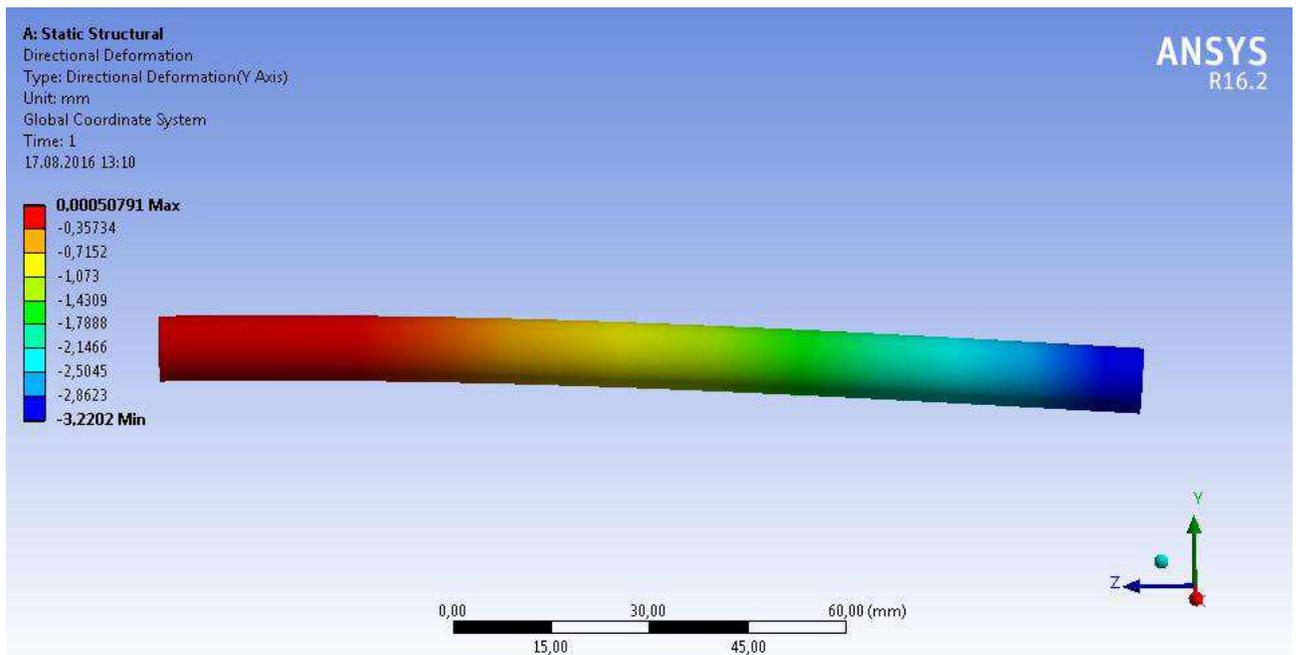
-152.7887

Ansys solution

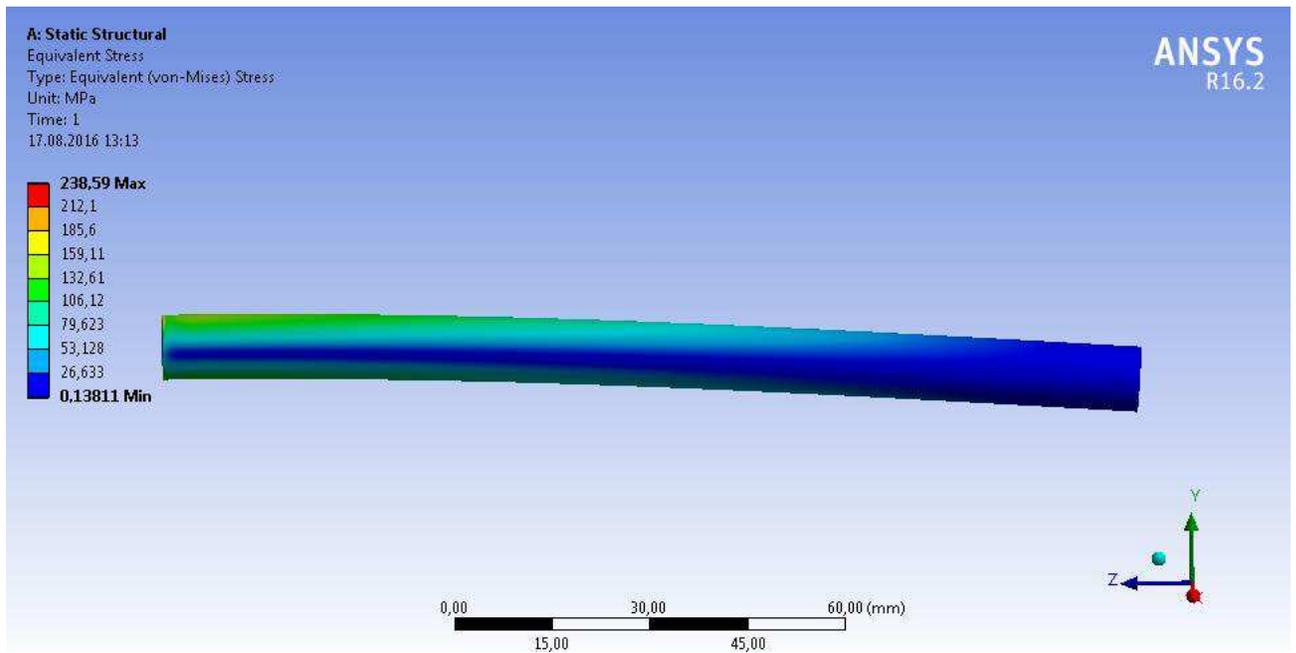
Boundary conditions:



Deflection:



Stress:



Results

	Maximal deflection, mm	Maximal stress, mm
Theoretical	3,32	152,8
MATLAB	3,32	152,8
ANSYS	3,22	159

Error:

	Deflection error, %	Stress error, %
MATLAB	0	0
ANSYS	0	4%

Discussion

As we see, we have very good correlation of results and very small error. The main reason we select 159 MPa for maximal stress instead of 238,59 (Ansys) is because the green line (159 MPa) is more uniformly distributed through the beam, when the red area (238,59 MPa) appears more rapidly and has smaller zone that can be caused by grid error. Total error is less than 4%, which is a very good result.



ASSIGNMENT. ESSAYSHARK

[Place free inquiry](#)

Submit instructions for free, pay only when you see the results.