Exercício Individual 6

Luiz Georg 15/0041390 Opção V

October 28, 2021

The problem values for option V were tabulated in Table 1, with retangular cross section of sides b and h. We want to determine the critical buckling load P_{CR} and the buckling direction on a column with simply supported ends, and compare the results for numerical and analytical methods.

Table 1: Values for Option V

Symbol	Value	Unit	Description
L	450	mm	Length of column
b	10	mm	Bottom flange length
h	18	mm	Web length
E	70	GPa	Modulus of Elasticity
ν	0.3	-	Poisson's coefficient

1 Analytical solution

The critical load P_{CR} can be calculated from Eq. (1), where I is the moment of inertia for the buckling direction and l_e is the equivalent length of the column. For our simply supported ends, $l_e = L$.

$$P_{CR} = \frac{\pi^2 EI}{l_e^2} \tag{1}$$

Since the column will buckle in the direction of least resistance, and the modulus of elasticity E is constant (isotropic material), we know buckling will happen around the minimum moment of inertia. For the rectangular cross-section, the minimum moment of inertia will happen around the axis of the longest dimension, and the direction of buckling will be perpendicular to that axis. For our case, the vertical axis is the axis of minimum inertia and the column will deform horizontally. The minimum moment of inertia is $I = b^3h/12 = 1500\,\mathrm{mm}^4$.

Substituting the values into Eq. (1), we find the critical buckling load $P_{CR} \approx 8459.7\,\mathrm{N}$.

2 Numerical solution

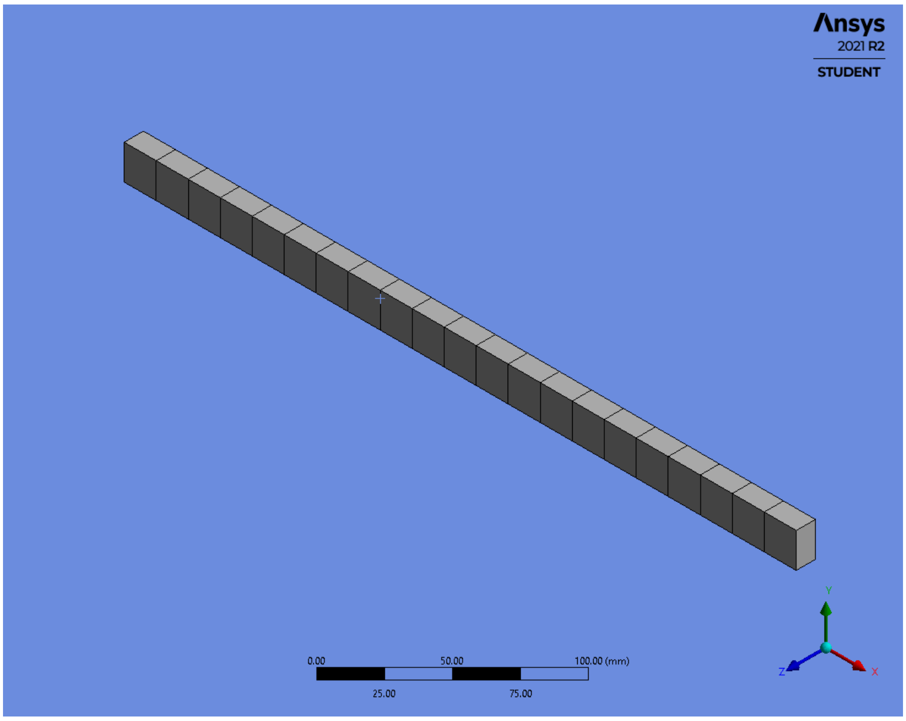
The buckling load can also be calculated numerically using finite element methods software *Ansys*. To solve the problem, we have to input the geometry (finite element mesh), the load case (boundary conditions and applied loads), and interpret the results. We have to use the *Static Structural* analysis option for a preload calculation, coupled with an *Eigenvalue Buckling* analysis for the actual buckling study.

2.1 Geometry

We can use the sofware *DesignModeler* inside *Ansys* to model our geometry and automatically mesh it. The column can be modeled by a *Line Body* of length L, with a rectangular cross-section of appropriate dimensions.

Inside the *Static Structural* model, we set our mesh *Element Order* to *Quadratic* and otherwise use the software's default values, obtaining the mesh shown in Fig. 1. In the 1-D case of the *Line Body*, the mesh does not need to be refined to generate accurate results.

Figure 1: Finite Element Mesh generated by the software



2.2 Load case

In the *Static Structural* analysis, we have to input the load case for the preload. Both ends of the beam element are simply supported; however, to apply the load on one end, we have to relax the support condition for that end along the direction of the load. Therefore, the support conditions on the base of the beam are of *Displacement*, set to 0 along all directions, while the loaded end has *Displacement* set to 0 only along the directions perpendicular to the beam length. To prevent rigid body motion, we also need to apply a *Fixed Rotation* support to any part of the geometry (as the beam won't be under any torsional loads, the placement isn't relevant and is only needed for numerical stability). For this simulation, the *Fixed Rotation* condition was applied to the base vertex. Finally, we also need the preload condition, which is a force applied to the end of the beam.

This analysis calculates the tension distribution across the body, and the *Eigenvalue Buckling* calculates the buckling load as a multiple of the preload. Therefore, we set the unitary load of 1 N to simplify calculations. The load case can be seen in Fig. 2.

We can then add an *Eigenvalue Buckling* analysis to the model, and set the *Pre-Stress* environment to the *Static Structural* analysis set up before. We can set *Max Modes to Find* to 1, as we are only interested in the first buckling mode.

2.3 Results

With the Geometry and the Load Case correctly set up, we can solve the model. We are interested in the magnitude of the critical buckling load P_{CR} and the direction of buckling θ. We can set the desired results to find the total deformation and the directional deformations perpendicular to the column axis. In any of the results, the *Load Multiplier* represents P_{CR}. We can use the values of directional deformation and basic trigonometry to find θ. It is important to note that buckling can happen to either side of the buckling axis.

The total deformation results can be seen in Fig. 3. The distribution of directional deformation is identical to the total deformation and, therefore, not relevant to show here; we are interested only in the maximum value in each direction. The buckling direction can be calculated as $\theta = \text{atan}(u_v/u_h)$, where u_v and u_h are the deformations in the vertical and horizontal directions, respectively, relative to the cross-section. The relevant values are shown in Table 2

Figure 2: Loads and Boundary conditions applied to the geometry

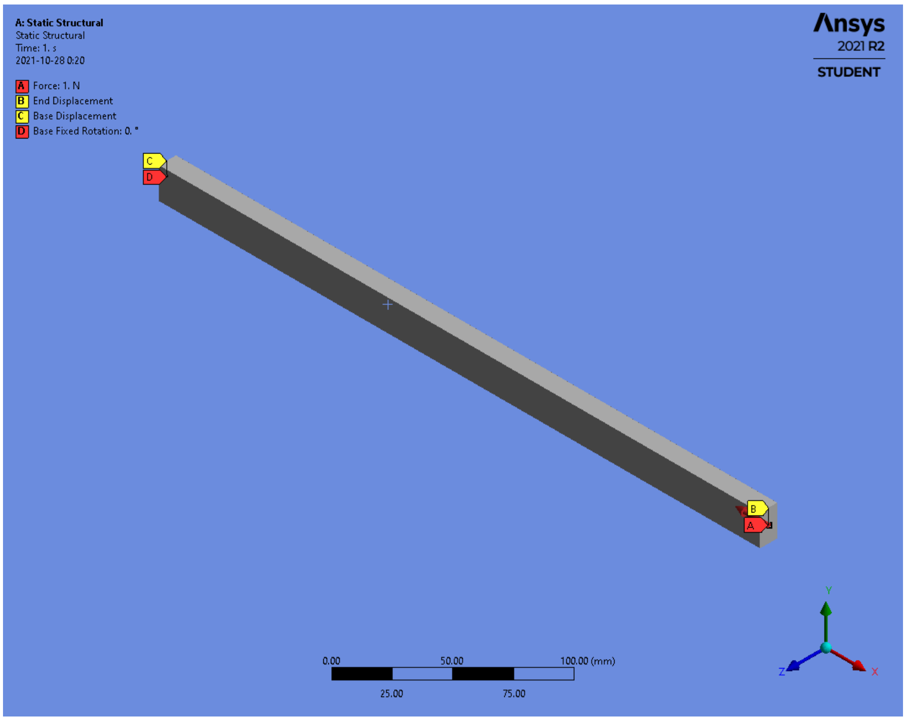
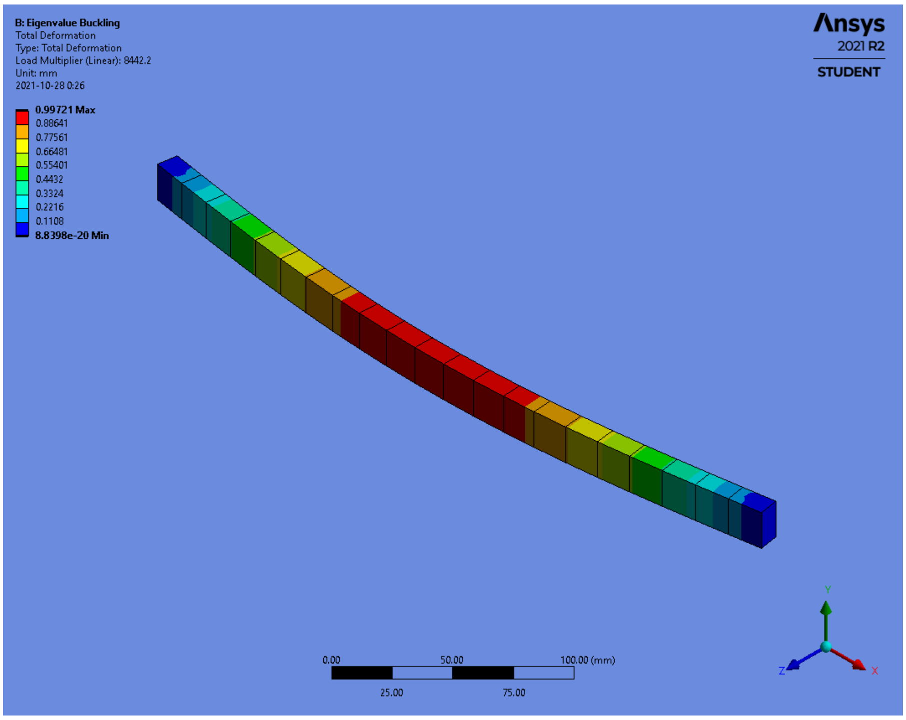


Table 2: Numerical results

P_{CR}	8442 N
u_v	0.9972 mm
u_h	−0.0000 mm
θ	−0.0000°

Figure 3: Total deformation across the column under critical buckling load.



3 Comparison of methods

Finally, we can compare the results found in Sections 1 and 2. We can also calculate the numerical error relative to the analytical approach. The comparison is shown in Table 3. The errors are extremely small, as to be negligible.

Table 3: Comparison of analytical and numerical results

	P_{CR}	θ
Analytical Method	8460 N	0°
Numerical Method	8442 N	−0.0000°
Relative Error	−0.0021 %	−0.0000 %