

COLUMN BUCKLING WITH VARIOUS BOUNDARY CONDITIONS

Overview

This simulation shows the workflow of conducting an eigenvalue buckling analysis on a slender column. A column is loaded under various boundary conditions, and the corresponding buckling load is obtained.

Goals

- To understand the workflow of the eigenvalue buckling analysis.

Steps

- Case1. Column buckling analysis with fixed-simply supported support.
 - Create a Static Structural system in Ansys workbench.
 - Import the column geometry
 - Mesh the column in Mechanical.
 - Fix the lower end of the column and constraint the translational displacement of the upper end. Apply a compression force of 1N to the upper end.
 - Run the simulation
 - Go back to Workbench. Drag an eigenvalue analysis to the solution of the static structural analysis.
 - Open the eigenvalue analysis in Mechanical. Run the eigenvalue analysis (no more boundary condition is needed) and obtain the buckling load. The buckling load for this case is 1.28e6N.
- Case2. Column buckling analysis with Simply-simply supported support.
 - Go back to Workbench. Duplicate the static structural analysis system and rename it with a proper name.
 - Drag an eigenvalue analysis to the solution of the static structural analysis.
 - Edit the boundary condition in Mechanical. The out-of-plane rotation degree of freedom needs to be constrained to prevent rigid body motion.
 - Run the analysis. The buckling load for this case is 6.31e5N.
- Case3. Column buckling analysis with Fixed-free support.
 - Replicate the same procedure as Case2, but modify the boundary condition to Fixed-free.
 - Run the analysis. The buckling load for this case is 1.58e5N.
 - Case3. Column buckling analysis with Fixed-Fixed support. Duplicate the process of step 3. The buckling load obtained is 2.48e6N.

Summary

This simulation demonstrates how to conduct an eigenvalue buckling analysis on a slender column. The effect of boundary conditions is also investigated.