Peter von Buelow
Winter 2022

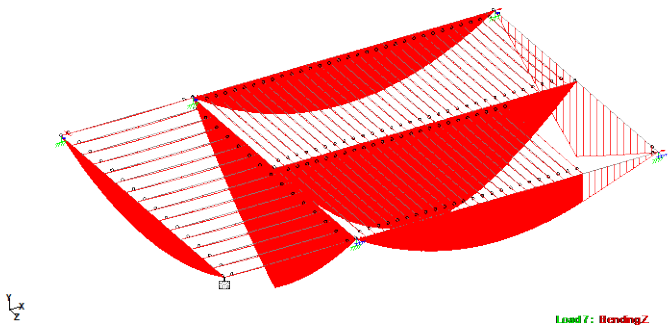
Intro to STAAD.Pro

Description

This project serves as an initial introduction to the use of finite element analysis with STAAD.Pro.

Objectives

- to develop a simple floor model in STAAD.Pro
- to analyze a basic and combination load conditions.
- to run an analysis.
- to define member properties and run code check.
- to design members sizes based on code requirements.
- to document post analysis results using STAAD Report.



Procedure

- Enter the data file of the floor system. Use `floor22.dxf` to import the geometry to the STAAD editor.
- Using the STAAD graphic input menus, define support conditions, member properties, and loadings (as on the tutorial handout). Use load combination of D+L+Lr. For roof: D=6psf, Lr=20psf. For floor: D=5psf, L=40psf.
- Run the analysis in STAAD.
- Look at (do not print) the results. If some members fail, increase them in size, but try to find the smallest size that will pass for each group of members.
- Use the Report feature to prepare:
    1. Picture showing joint and member numbers and supports.
    2. Picture showing the deflections.
    3. Picture showing the moments (Mz).
    4. List Support Reactions
    5. Section Properties
    6. Node Displacement Summary
    7. Beam Force Detail Summary
    8. Full Section plot
    9. Utilization Ratio (list and plot)

Report

- Cover sheet with name, date, project, etc.
- Input file (final version of .std file)
- STAAD report with plots and printed output (steps 1-8)

Due Date 19 March, 2022