ETH
Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich

ibk
Institute of Structural Engineering

# Assignment #1: Structural Analysis by Direct Stiffness Method

# Method of Finite Elements I

## 0. Problem statement:

Use the MATLAB code and Demo provided on the course website for simulating the 2D structure, illustrated in Fig.1, by assuming the following parameters:

| **Properties:** | | **Loadings:** | |
|---|---|---|---|
| Material: | Steel S235 | $P_1$ = | 200 kN |
| Beam Elements: | HEB 260 | $P_2$ = | 500 kN |
| Truss Elements: | ROR 193.7 • 25.0 | $p_1$ = | 20 kN/m |
| Spring Element: | k = 50*$10^6$ Nm/rad | $p_2$ = | Top: +40°$^C$, Bottom: +20°$^C$ |

The bridge deck is warmed by the sun, resulting in an increase of 40°C on the top and 20°C at the bottom of the deck. The triangular construction above the bridge is used for the free cantilever method (Freivorbau). The loads $P_1$ and $P_2$ are point loads, while $p_1$ and $p_2$ are distributed element loads.

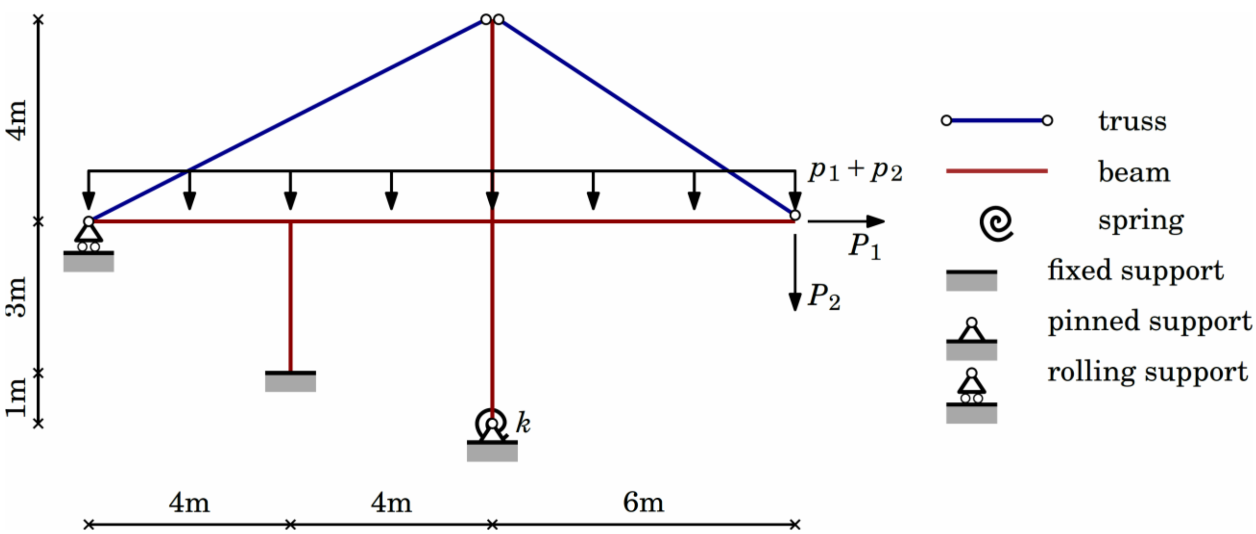


Fig. 1: 2D Frame Structure

## 1. Modeling assumptions

State the modeling assumptions underlying this analysis, i.e. for which kind of analysis will the results calculated using this method be valid?

## 2. Preparing the numerical model

Prepare the numerical model so that it may be used as input for the provided Matlab code. Do so by discretizing the physical model into a numerical model implementing the steps stated below.

- Introduce a global coordinate system.
- Assign nodes and nodal coordinates.
- Assign elements with starting and ending nodes including orientation.
- Calculate the total number of unknowns of the system given the assigned nodes.
- Divide the number of unknowns into degrees of freedom (DOF) and restraints due to the boundary conditions.
- Assign unique identification numbers to each of the unknowns at the nodes using the scheme described in the pdf-document accompanying the provided Matlab code.
- By hand, create the matrix of unique identification numbers by adding the contributions of each element.
- Prepare the remaining input (such as node coordinates, element properties, loading etc.) to match the structure in Fig. 1.
  **Attention: Be sure to use a consistent set of units!**

## 3. Augmenting the direct stiffness method and performing the analysis in Matlab

In order to perform the analysis, please find the local stiffness matrix for a truss in your class notes. The local stiffness matrix for a spring is that of a truss however with entries equal to the spring stiffness k instead of the truss stiffness EA/L. Derive the local truss element mass matrix using an approach consistent with the corresponding shape functions **N**. Additionally, derive the loading vector for the temperature loading as detailed in the homework intro session.

$$\boldsymbol{m}_{element}^{local} = \rho A \int_{x=0}^{L} \boldsymbol{N}^{\boldsymbol{T}} \boldsymbol{N} dx \quad \text{and} \quad \boldsymbol{f}_{element}^{local} = \int_{x=0}^{L} \boldsymbol{N}^{\boldsymbol{T}} \boldsymbol{f} dx = \boldsymbol{k}_{element}^{local} * \boldsymbol{u}_{element}^{local}$$

## 4. Checking the validity of the modeling assumptions

Check the validity of your analysis assumptions in an approximate fashion by plotting the moment, shear and normal force diagrams for the structure.

- Does the structure yield under these loads?
- What would possible consequences be?
- In a brief, qualitative manner, how would you further proceed to capture this in a numerical analysis?

## 5. Calculating support reactions

Extend the provided Matlab code to calculate support reactions. Explain briefly how you accomplished this and at what position in the provided Matlab code you made changes.

## 6. Validation using an alternate structural analysis program

Program the same structure using a structural analysis software of your choosing (Statik, SAP2000, etc.). You may use the relevant demo, available on the class website, and modify it accordingly for performing the analysis. Compare the obtained nodal displacements results and modal frequencies to the ones of your Matlab code.

## 7. Parameter Study

A colleague falsely criticizes some of your modelling choices:

1. Due to the special support conditions of the longest column, its length is in fact 4.3m.
2. This impacts the results more than adding a torsional spring at the base, as a common modelling choice is to assign a fixed support in such cases anyways.

By means of a parameter study for various column lengths and ranges of support fixation show your colleague his error. Do so by reporting the maximal moments, shear and normal forces for each combination and additionally report if any elements fail (SIA 263). You may omit stability if you wish.

## 8. Extra Credit

Modify the 6 DOF beam element stiffness matrix such that a (moment) joint is incorporated on one side. Does this also affect the element loading vector?

**Hints:** (1) Write out the six relations governed by KU=F forming a system of linear equations.
(2) Set the one (moment/rotation) reaction equation =0 as its contribution is released.
(3) Solve this equations for the DOF corresponding to the previously released reaction.
(4) Back substitute this into (1).
(5) Reorder and form the modified stiffness matrix.

Show that by implementing joints on both ends of a 6 DOF beam element you can derive the truss element stiffness matrix.

**In general literature this is termed a “member end release”.**

## 9. Tips:

Modify the Matlab functions that retrieve the mass and stiffness matrix so that they become similar to the element loading one. It is simplest to pass a variable which defines the type of element (beam of truss) to return. A switch statement is usually the easiest way to accomplish this. Naturally this additional variable must be specified somewhere at some point in the input.

## Administrative matters

Submit your report, including the utilized MATLAB code, via email to egger (at) ibk (dot) baug (dot) ethz (dot) ch. Please remember to also list all you group members!