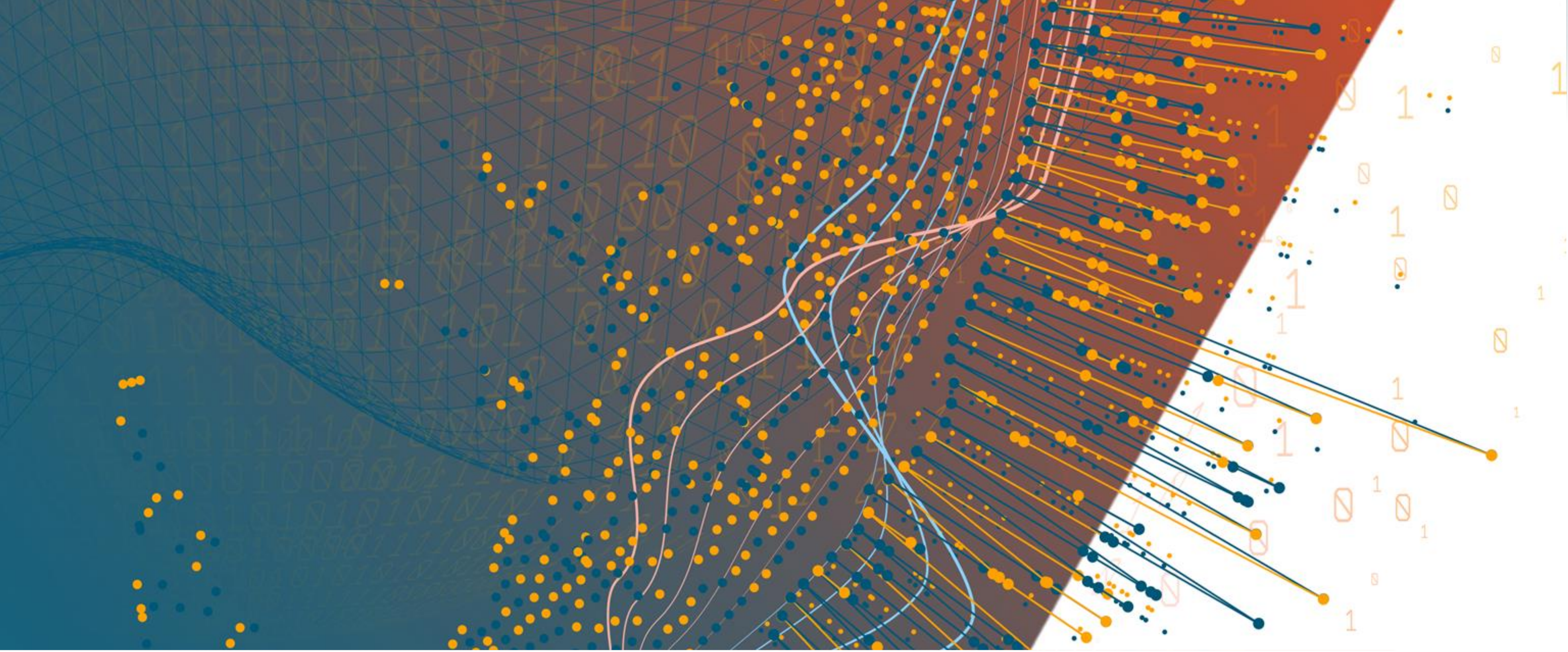


ALTAIR

ALTAIR STUDENT WEBINAR SERIES - FINITE ELEMENT MODELLING WITH HYPERWORKS™ NEXT GENERATION

Marius Müller / Altair Ambassador / September 17, 2021

Speakers profile

Altair Student Webinar Series 2021

Speaker Profile

- **Bachelor's degree** in Mechanical Engineering from Graz University of Technology
- **Altair Ambassador** since October 2018
- FEA-Consultant of **TU Graz Racing Team**
- Former team principal of **TERA TU Graz**
- Part time **Project Collaborator** (FEA Engineer) at Institute of Materials Science, Joining and Forming, Working group Tools & Forming
  Graz University of Technology
- Part time **FEA Engineer** at **PJ Messtechnik GmbH**
  (https://pjm.co.at/en/; https://at.linkedin.com/company/pjm)

Marius Müller

Altair Ambassador

ALTAIR

Agenda

Agenda

Altair Student Webinar series - Finite Element Modelling with HyperWorks Next Generation

- Crash Course Finite Element Analysis
- What is HyperWorks™?
- Basic interaction with HyperWorks™ 2021
- Geometry
- 1D meshing and Analysis

ALTAIR

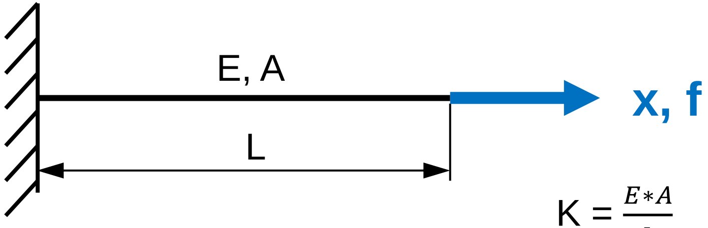
Crash Course Finite Elements

Crash Course Finite Elements

Basics of Finite Element Analysis

x … displacement vector
f … external force vector
K… Tension spring stiffness

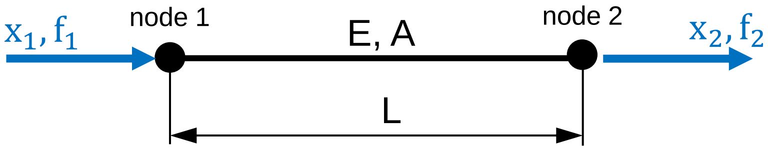
Hooke's law:
K***x** = **f**



$K = \frac{E*A}{L}$

E… E Modulus
A… Cross section
L… Length of the rod

Basic definition of a rod



Equilibrium of forces

$$\left.\begin{aligned} f_1 &= K*x_1 - K*x_2 \\ f_2 &= K*x_2 - K*x_1 \end{aligned}\right\} \begin{bmatrix} K & -K \\ -K & K \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}$$

$$\begin{bmatrix} K & -K \\ -K & K \end{bmatrix} = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix}$$

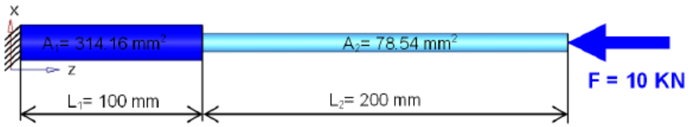
… Element stiffness matrix for a horizontal rod

ALTAIR

Crash Course Finite Elements

Basics of Finite Element Analysis – An example 1/2

We can break down the loaded bar setup computationally as follows:



Material Steel:

E =210 GPa

1. FE Model

2. Element Matrix

$$\mathbf{K} = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix}$$

3. Global Matrix

$$\mathbf{K}_1 = \begin{bmatrix} \frac{210*314.16}{100} & -\frac{210*314.16}{100} \\ -\frac{210*314.16}{100} & \frac{210*314.16}{100} \end{bmatrix} = \begin{bmatrix} 659.74 & -659.74 \\ -659.74 & 659.74 \end{bmatrix} \qquad \mathbf{K}_2 = \begin{bmatrix} \frac{210*78.54}{200} & -\frac{210*78.54}{200} \\ -\frac{210*78.54}{200} & \frac{210*78.54}{200} \end{bmatrix} = \begin{bmatrix} 82.47 & -82.47 \\ -82.47 & 82.47 \end{bmatrix}$$

$$\mathbf{K}_G = \begin{array}{c|ccc|} & 1 & 2 & 3 \\ \hline 1 & 659.74 & -659.74 & 0 \\ 2 & -659.74 & 659.74+82.47 & -82.47 \\ 3 & 0 & -82.47 & 82.47 \\ \hline \end{array}$$

4. Forces and Displacements

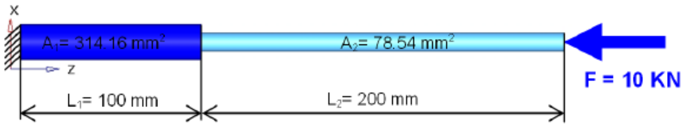
$$\mathbf{f} = \begin{Bmatrix} 0 \\ 0 \\ -10 \end{Bmatrix} \qquad \mathbf{x} = \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix}$$

ALTAIR

Crash Course Finite Elements

Basics of Finite Element Analysis – An example 2/2

By combining (5) the global system with (6) the prescribed DOF, (7) the system can be solved for strain, stress, and forces.



Material Steel:

E =210 GPa

5. Global System

$$\begin{bmatrix} 659.74 & -659.74 & 0 \\ -659.74 & 742.21 & -82.47 \\ 0 & -82.47 & 82.47 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -10 \end{Bmatrix}$$

6. Eliminate the Prescribed DOF

$$\begin{bmatrix} 742.21 & -82.47 \\ -82.47 & 82.47 \end{bmatrix} \begin{Bmatrix} x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10 \end{Bmatrix}$$

7. Solving the system

$$\begin{Bmatrix} x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} -0.0152 \\ -0.1364 \end{Bmatrix}$$

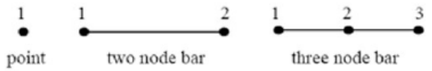
Strain	Stress	Forces
$\varepsilon_1 = \frac{\Delta L}{L} = \frac{x_2 - x_1}{L_1} = \frac{-0.0152 - 0}{100} = -1.52 * 10^{-4}$ mm/mm	$\sigma_1 = E\varepsilon_1 = -1.52 * 10^{-4} * 210$ GPa $= -0.032$ GPa	$f_1 = \sigma_1 A_1 = -0.032 * 314.16$ kN $= -10$ kN
$\varepsilon_2 = \frac{\Delta L}{L} = \frac{x_3 - x_2}{L} = \frac{-0.1364 - 0.0152}{200} = -6.06 * 10^{-4}$ mm/mm	$\sigma_2 = E\varepsilon_2 = -6.06 * 10^{-4} * 210$ GPa $= -0.127$ GPa	$f_2 = \sigma_2 A_2 = -0.127 * 78.54$ kN $= -10$ kN

Crash Course Finite Elements

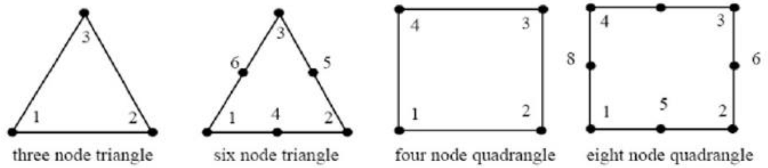
Element Types

In OptiStruct™, there are a selection of element types available. They are usually categorised as 1D, 2D and 3D elements.

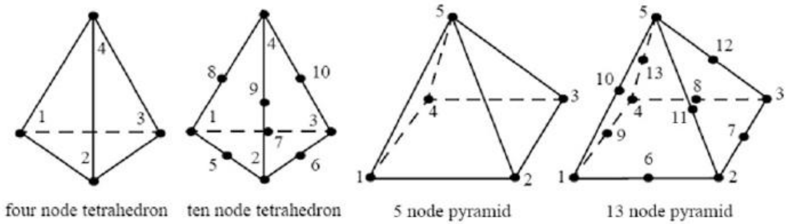
- 1D ore line elements



- 2D or area elements



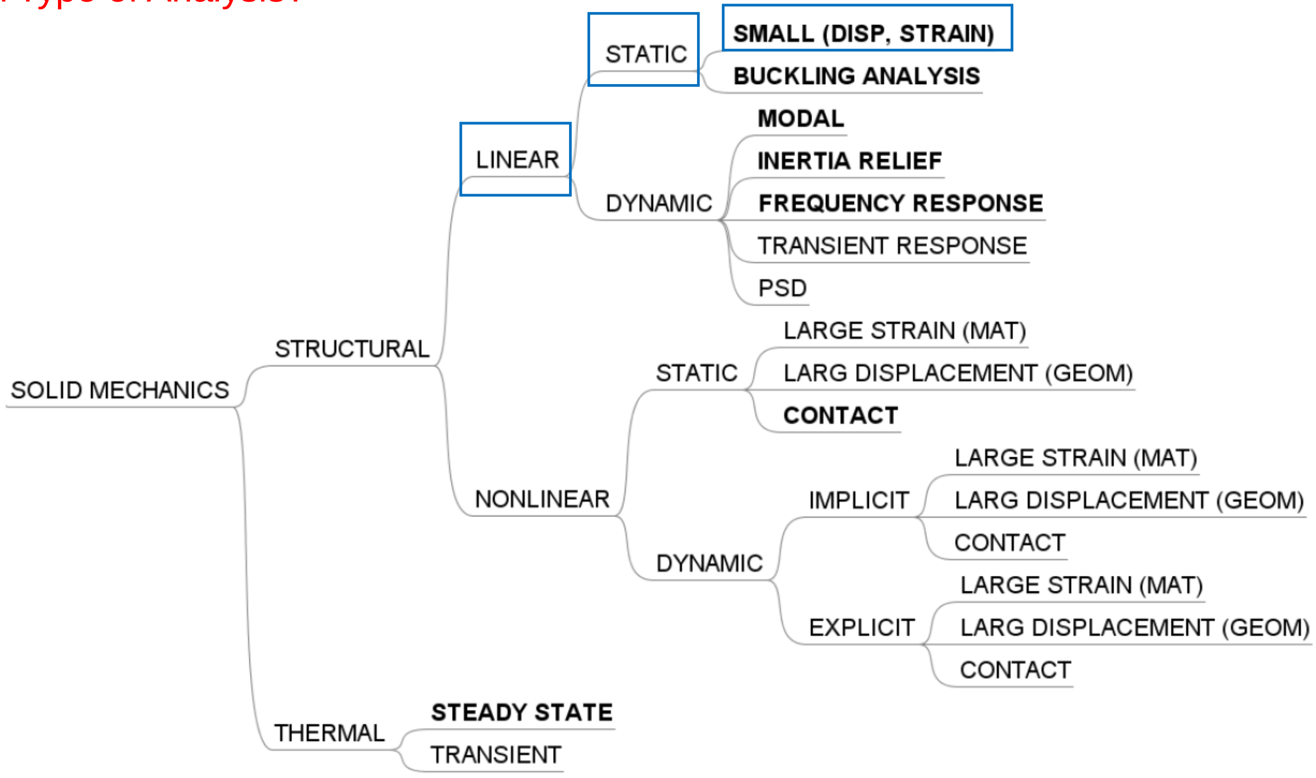
- 3D or volume elements



ALTAIR

Crash Course Finite Elements

Which Type of Analysis?



ALTAIR

Crash Course Finite Elements

Linear Static Analysis

In mechanics the static state is defined as the state of a system that is in equilibrium under an action of balanced forces and moments so that they remain at rest (no velocity). A linear analysis assumes that the loading causes negligible changes to the stiffness of the structure.

- All deformations and strains are small (in the elastic range).
- Stresses are assumed to be linear functions of the strains.
- Structural deformations are proportional to the loads applied. This infers that the loading pattern does not change do to the deformed shape and no geometric stiffening occurs due to the application of the load.
- Material behaviour is a linear elastic one. Therefore, the material deforms along the straight line portion of the stress-strain curve (no plasticity or failures occur).
- No abrupt changes in stiffness such as two bodies come into or out of contact.

ALTAIR

What is HyperWorks™

What is HyperWorks™

Driving More Design with Simulation

HyperWorks™ provides:

- Easy-to-learn, effective workflows.
- Intuitive direct modelling for geometry creation and editing, mid-surface extraction, surface and midmeshing and mesh quality correction.
- Efficient assembly management.

HyperWorks™ offers:

- A complete environment to visualise, query and process result data.
- Gives access to a wide range of CAE data formats.
- Enabling full post-processing.

https://www.altair.com/hyperworks/

THANK YOU

altair.com

#ONLYFORWARD

ALTAIR