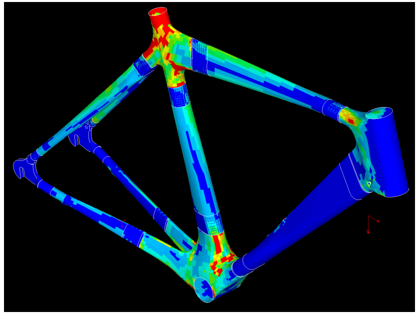
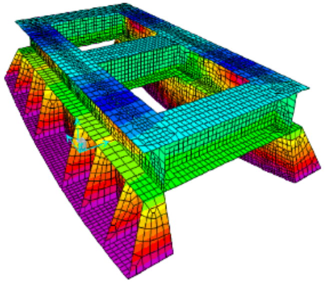


(Credit: Form Lab)



(Credit: Adesol)

Introduction

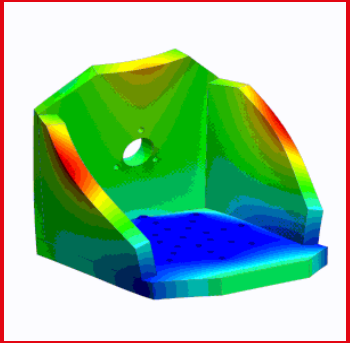
General information

ME-473

Dynamic finite element analysis of structures

Stefano Burzio

Welcome!



(Source: Sentek)

- Who are the instructors?
- How is the learning process structured?
- What reading materials are recommended?
- What does the curriculum include?
- How will my learning be assessed?
- Why is dynamic FEA important?

ME-473 Teaching staff

Lecturer

Stefano Burzio

Scientific collaborator

Teaching assistant

Adam Lecroart

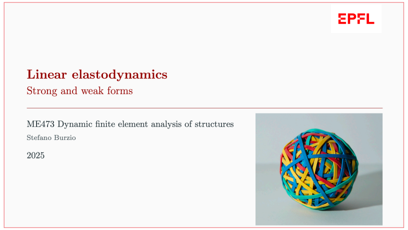
Master's student in Mechanical Engineering

Who am I?

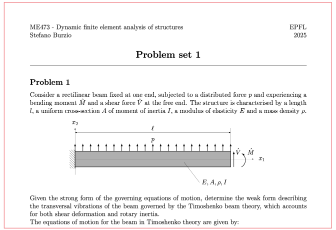
Lecture	Tuesday 14h - 16h	GCD 0386
Exercise	Tuesday 16h - 17h	GCD 0386
Office hours	Thursday 11h - 12h	ME A2 390

Moodle page: (Link)

An official platform for announcements and sharing materials, including lecture notes, exercises, practice problems, and more.



Slides



ME473 - Dynamic finite element analysis of structures
Stefano Burzio
EPFL
2025

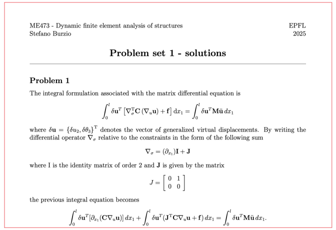
Problem set 1

Problem 1

Consider a rectilinear beam fixed at one end, subjected to a distributed force p and experiencing a bending moment $\bar{M}$ and a shear force $\bar{V}$ at the free end. The structure is characterised by a length l, a uniform cross-section A of moment of inertia I, a modulus of elasticity E and a mass density ρ.

Given the strong form of the governing equations of motion, determine the weak form describing the transversal vibrations of the beam governed by the Timoshenko beam theory, which accounts for both shear deformation and rotary inertia.
The equations of motion for the beam in Timoshenko theory are given by:

Problem sets



ME473 - Dynamic finite element analysis of structures
Stefano Burzio
EPFL
2025

Problem set 1 - solutions

Problem 1

The integral formulation associated with the matrix differential equation is

$$\int_0^l \delta \mathbf{u}^T \left[\nabla_e^T \mathbf{C}(\nabla_e \mathbf{u}) + \mathbf{f}\right] dx_1 = \int_0^l \delta \mathbf{u}^T \mathbf{M}\ddot{\mathbf{u}}\, dx_1$$

where $\delta\mathbf{u} = \{\delta u_2, \delta\theta_3\}^T$ denotes the vector of generalized virtual displacements. By writing the differential operator ∇_e relative to the constraints in the form of the following sum

$$\nabla_e = (\partial_{x_1})\mathbf{I} + \mathbf{J}$$

where $\mathbf{I}$ is the identity matrix of order 2 and $\mathbf{J}$ is given by the matrix

$$J = \begin{bmatrix} 0 & 1 \\ 0 & 0 \end{bmatrix}$$

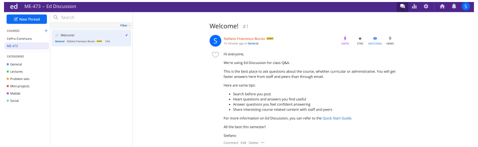
the previous integral equation becomes

$$\int_0^l \delta \mathbf{u}^T [\partial_{x_1}(\mathbf{C}\nabla_e \mathbf{u})]\, dx_1 + \int_0^l \delta \mathbf{u}^T (\mathbf{J}^T \mathbf{C}\nabla_e \mathbf{u} + \mathbf{f})\, dx_1 = \int_0^l \delta \mathbf{u}^T \mathbf{M}\ddot{\mathbf{u}}\, dx_1.$$

Problem solutions

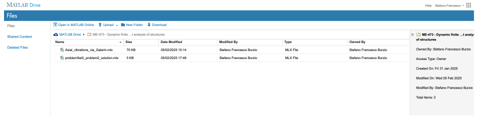
Ed discussion forum: (Link)

A general student forum, where both lecturers and fellow students can answer your questions.

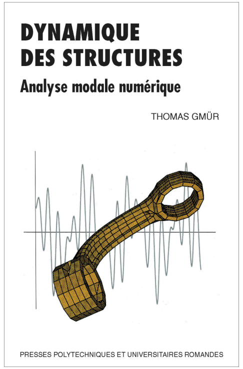


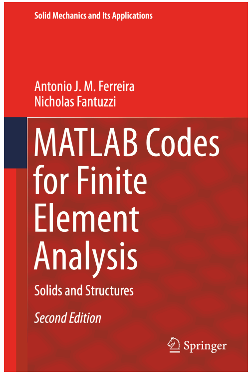
Matlab drive (Link)

Read-only online MATLAB code repository.
Log in with your Switch-edu account



Recommended books

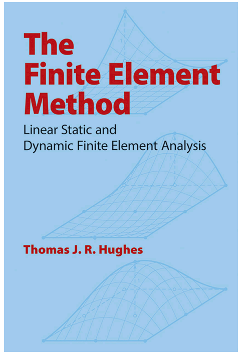


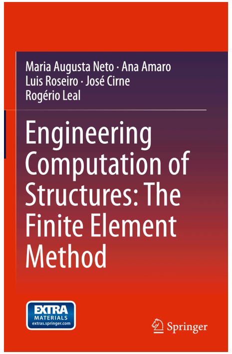


Main references:

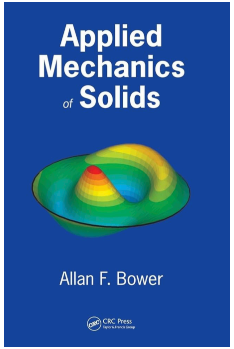
- Thomas Gmür
  Dynamique des structures (PPUR)
  EPFL library: (07 534 GMU 2012)
- Antonio Ferreira and Nicholas Fantuzzi
  MATLAB Codes for Finite Element Analysis
  Available online (link)

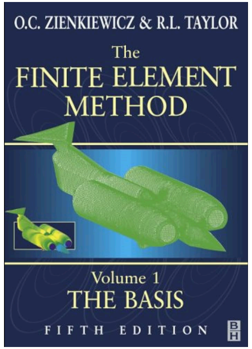
English

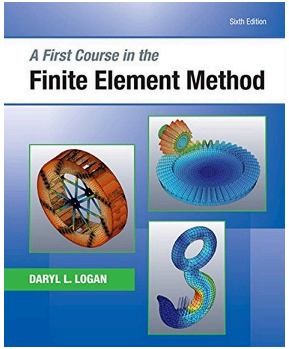




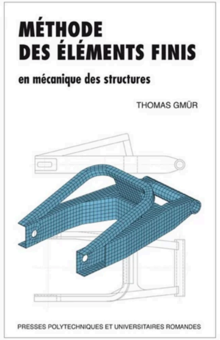
All-time classics







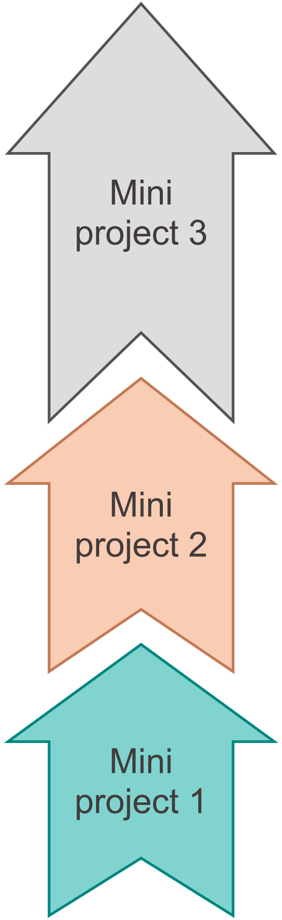
ME-372



Syllabus

Week	Module	Lecture topic	Problem set	Mini projects
1	Linear elastodynamics	Strong and weak forms	1	
2		Galerkin method	2	
3		Finite element method	3	Groups formation
4		Systematisation of the procedure	4	Project 1 statement
5		Solid elements and numerical integration		
6	Special structural elements	Trusses	6	
7		Planar beams	7	**Project 1 submission**
8		Frames and grids	8	Project 2 statement
9		Kirchoff plates	9	
10		Riessen-Mindlin plates	10	
11	Analysis of free and forced vibrations	Lancoz, subspace algorithms	11	**Project 2 submission**
12		Gyroscopic and dissipative systems	12	Project 3 statement
13		Newmark schemes	13	
14		Wilson schemes		

Mini projects submission guidelines



- **Objective**: put course content into practice
- **Groups**: 3 to 4 students (register groups on Moodle)
- **Group assessment**:
  - work divided equally among the group
  - evaluation criteria on Moodle
- **Deliverables**:
  - PDF report (explaining your analysis and detailing the code)
  - Simulation file (MATLAB, ANSYS, Abaqus)
- **Deadlines**:

Project	Topic	Statement	Submission
1	Modal analysis of a solid	September 30	**October 31**
2	Modal analysis of a structure	November 4	**November 28**
3	Transient analysis	December 2	**January 9**

Main learning objectives

1. **Understand the fundamentals of dynamic FEA**

   Develop a strong theoretical foundation in the principles of dynamic analysis, including *modal analysis* and *frequency response*.

2. **Analyse the dynamic behaviour of structural components**

   Study the dynamic response of specific structures such as *rods*, *trusses*, *beams*, *plates*, and *shells* under various loading and boundary conditions.

3. **Develop proficiency in finite element formulation for dynamic problems**

   Learn to derive and implement finite element equations for time-dependent structural dynamics problems, including damping and rotational effects.

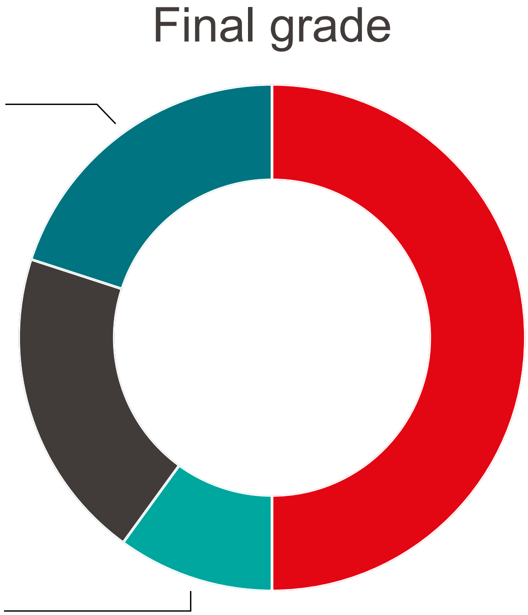
4. **Apply computational methods for solving dynamic systems**

   Utilise numerical techniques such as direct integration, *modal superposition*, and *time-stepping methods* to solve complex dynamic problems.

5. **Implement advanced applications in engineering design**

   Apply dynamic FEA to real-world engineering problems, such as vibration analysis and linear dynamic response to dynamic loads, using commercial software.

Assessment method



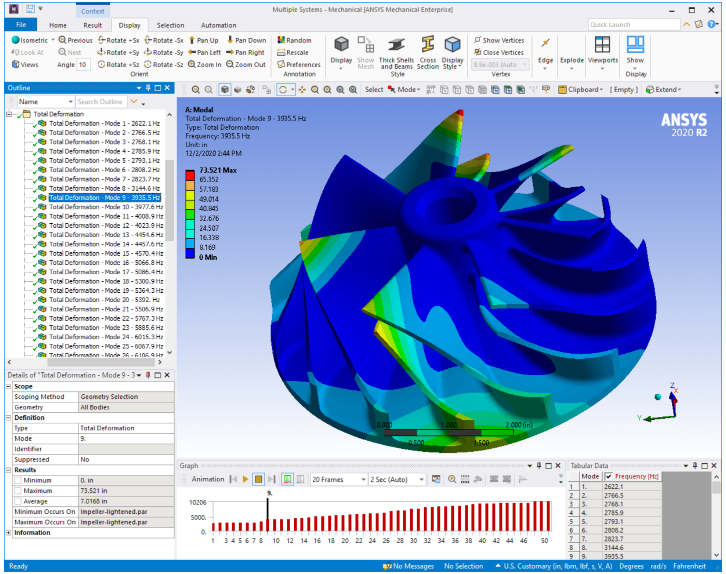
The finite element analysis dichotomy

Code developers

```
static public void verify()
{
    cgiSolverBlazeClass solver = new cgiSolverBlazeClass();
    solver.createStructure();

    StringBuilder sb = new StringBuilder();
    solver.getDefaultTestPath(ref sb);
    string sInputFileName = sb.ToString() + "\\tests\\newModels\\Verify-Example4-Sparse.r3a";

    // the following should be called after createStructure()
    cgiSolverBlazeClass.ListMessageDelegate ListMsg = new cgiSolverBlazeClass.ListMessageDelegate(Callback);
    cgiSolverBlazeClass.StatusMessageDelegate StatusMsg = new cgiSolverBlazeClass.StatusMessageDelegate(StatusCallback);
    cgiSolverBlazeClass.SparseSolverProgressDelegate SparseMsg = new cgiSolverBlazeClass.SparseSolverProgressDelegate(MlkProgress);
    solver.setListMessageFunction(ListMsg);
    solver.setStatusMessageFunction(StatusMsg);
    solver.setSparseSolverStatusFunction(SparseMsg);

    solver.setModelType((int)cgiModelEnum.kModel_Frame2D);

    // LENGTH=ft;   DIMENSION=in; FORCE=kip; FORCE_LINE=kip/ft; MOMENT=kip-ft; FORCE_SURFACE=lb/ft^2;
    // DISPLACEMENT_TRANS=in; DISPLACEMENT_ROTATE=rad; MODULUS=kip/in^2; WEIGHT_DENSITY=lb/ft^3; STRESS=lb/in^2
    // SPRING_TRANS_1D=lb/in; SPRING_ROTATE_1D=lb-in/rad; SPRING_TRANS_2D=kip/in^2; SPRING_TRANS_3D=kip/in^3
    solver.setStandardEnglishUnits();

    // define materials
    List<cgiMaterialCli> listMat = new List<cgiMaterialCli>();
    cgiMaterialCli mat = new cgiMaterialCli();
    mat.setId(1);
    mat.setProperties("Default222", 29000, 0.3, 450);
    listMat.Add(mat);
    solver.setMaterials(listMat);

    // define sections
    List<cgiSectionCli> listSect = new List<cgiSectionCli>();
    cgiSectionCli sect1 = new cgiSectionCli();
    sect1.setId(1);
    sect1.setProperties("W27X84", 24.8, 12.282, 12.7488, 2850, 106, 2.81);
    listSect.Add(sect1);
    cgiSectionCli sect2 = new cgiSectionCli();
    sect2.setId(2);
    sect2.setProperties("W27X84", 24.8, 12.282, 12.7488, 2850, 106, 2.81);
    listSect.Add(sect2);
```

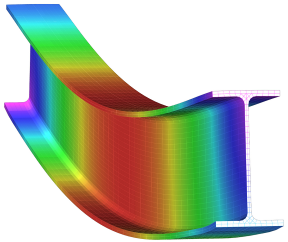
(Source: Cg-inc)

Tool users

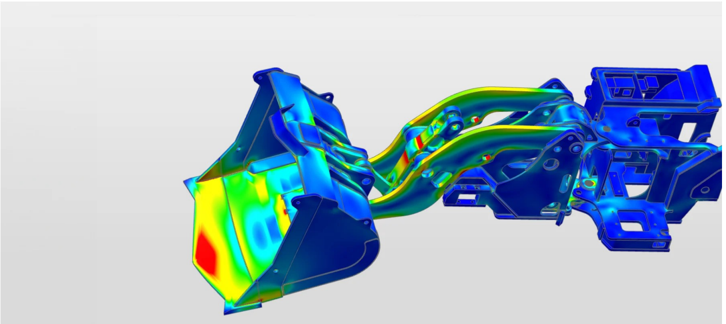


(Source: Padt)

Static analysis



(Source: Ansys)

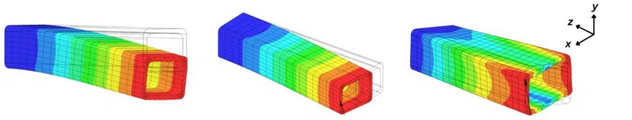


(Source: Rfamec)

Static analysis determines deformations due to constant loads.

- No matter how you apply the load, the solver assumes it happens infinitely slowly, meaning the loading method has no impact on structural behaviour.
- A constant load doesn't mean a linear response, buckling, yielding, and other nonlinear effects can occur, but the analysis remains static.

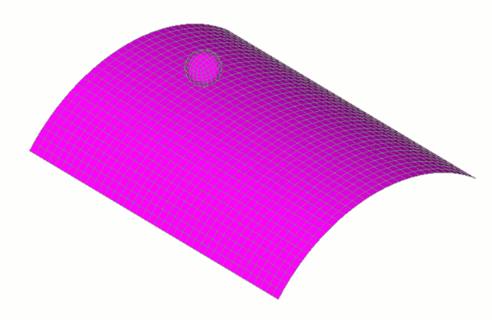
Dynamic analysis



(Source: Caeflow)

Modal analysis studies dynamic properties in the frequency domain.

- Natural frequencies are fundamental property in structural design; lower natural frequencies lead to larger displacement and thus greater risk.
- Resonance occurs when external excitation coincides with a natural frequency and can cause catastrophic failure.



(Source: Enterfea)

Transient analysis examines time-dependent structural responses to time-dependent excitations.

- Transient analysis helps predict peak stresses, displacements, and vibrations during short-term dynamic events such as impacts, blasts, or sudden load changes.

(Source: YouTube)