CNSTH421 Structural Analysis and Finite Elements

Electromechanical engineering project task description

Lab responsible:	Peter Berke	2025-2026 academic year
Teaching assistants:	Louis Remes & Noémi Mertens	

I- THE GOALS OF YOUR PROJECT

The finite element (FE) method is a computational tool used to solve problems governed by PDEs. Here it is applied to linear solid mechanics. The project work serves as a practical illustration of main concepts in the field of FE modeling with the following main goals:

1. to provide a *deep understanding of the theoretical concepts* of the method by their practical implementation into a FE code,
2. to offer a *first experience using a commercial finite element software* (pre-processing, analysis, post-processing),
3. to build up practical knowledge and experience for the *construction of a numerical model for a complex real-life problem* in your field,
4. to help *develop a critical mind* with respect to the results issued from the computational analysis.

II- ORGANIZATION OF YOUR WORK

Guidance for your project is provided for **six on campus lab sessions of four hours**, following the online labwork schedule. **Student participation** at these labworks **is compulsory and recorded**. Two unjustified absences from labworks imply being dissociated from the group and dealing with the project assignment in individual work. Unjustified absence from all of the labworks automatically results in a 0/20 grade for the project.

The project work is performed in **mixed ULB-VUB-International groups** of three students (when possible), defined before the first session. The resources of your group need to be used wisely in order to tackle all project related tasks, including: data gathering, Python coding, definition of the geometry, loads and boundary conditions, creation of the numerical model, convergence study, etc.

Note that **your project will require you to work outside of the guided sessions**. Python used for the programming part is an easily accessible tool, and the industrial finite element software, Salome-meca, can be downloaded free of charge for Linux and Windows. The Linux version is developed for Ubuntu-Linux, tested and confirmed to be stable using Ubuntu OS, while glitches may appear as a function of your hardware/software environment under Windows. Installation guides are provided for both OS. An ample base of tutorials is available online that is advised to be used together with the software documentation for Salome-meca.

III- Schedule of the Practical Sessions

The guidance given at the labwork sessions will follow the schedule below. For the sake of efficiency, **you must ensure having read the part of the course dedicated to the topic of the session and having assisted the corresponding short video beforehand**. Your project can be divided into two main parts and nine basic tasks (numbered below).

Part I – Sessions 1-3: FE implementation

1. General presentation of the project, introduction to the Python code,
2. *Element level implementation:* TETRA4 finite element routine: stiffness matrix of the element, computation of stresses and strains,
3. *FE level verification:* application of procedures that allow concluding on the correct functioning of the TETRA4 routine you coded in task 2 (local variable check and nodal displacements vs. generated strain and stress),
4. *Structural level implementation:* assembly of the structural stiffness matrix, enforcing boundary conditions, computing the reaction force vector,
5. *FE code verification:* application of procedures that allow concluding on the correct functioning of your code (e.g. patch tests),
6. *Feed-back on your project sheet.*

Part II – Session 4: Introduction to the industrial FE tool (Salome-meca)

7. Pre-processing (creating a CAD geometry, CAD geometry to FE mesh, application of materials and BC), FE analysis and graphical post-processing (visualization of the deformed shape, stress and strain tensors, cut views, etc.),
8. *Solving benchmark problems:* application to a practical problem in solid mechanics, including a mesh convergence study using a set of different mesh refinements.

Part II – Session 4 to 6: Project work 9. Group work, guidance available on demand. This part includes running a simulation of your project with the commercial FE software and <u>with your FE code</u> and the comparison of the results with the commercial code.

IV- Deliverables

Session 2: an *Identification Sheet* of your project has to be handed in (one A4 sheet single-sided printing).

The Identification Sheet is a *sketch of your project problem* containing: (i) the chosen geometry, (ii) the material properties, (iii) the estimation of loads, (iv) how the boundary conditions are applied, (v) how to adjust your model compared to the experimental results. You will receive a constructive feed-back on your project sheet to help you start efficiently your project assignment.

19th December 2025 at 1 PM: <u>final</u> deadline for receiving the *Project Report in PDF format* by email (peter.berke@ulb.be). *Your zipped Python code and Salome-meca <u>input files</u> must accompany the report*. **If the report and these compulsory attachments are not in the inbox by this deadline the score is automatically set to 0/20.**

Note that you can also use file sharing applications in case the attachments are too voluminous for sending them by email. Please avoid sending the Salome-meca result files,

unless there is a specific, exceptional reason for this, which you are expected to elaborate in your report.
It is emphasized that a **systematic cross-check** will be performed between the submitted **Python codes** of the different groups and in case of a **too close match a penalty of 0/20 to the Python assignment may be applied to both groups**.

Note that the **use of generative artificial intelligence as a supportive tool is limited** to: improving the structure of your report (respecting the predefined main structure), making your wording clearer, correcting grammar and enhancing the vocabulary. The use of generative artificial intelligence **for the coding assignment is not allowed**. You must respect 1) transparency: expose clearly how and which AI tool was used and justify the need for this in section 5 of the report; 2) responsible use: authors' rights and personal data must be respected. **The use of generative AI in the project tasks beyond what is allowed implies automatically the academic sanction of 0/20 for the project assignment.**

V- FINAL REPORT

This document retraces your project work with **a proper selection and presentation of the most relevant information**. The main objective is to show the relevant modeling choices you made and to demonstrate your critical mind with respect to the computational results and the modeling assumptions.

The form of the final report must respect:
- Font : Times New Roman
- Font size : 12
- Lines spacing : 1.5 lines
- Margin (top, bottom, left, right) : 2 cm
- Text alignment : justified
- Sections 1-4 are a total of 7 single-sided pages of text <u>excluding figures</u>.

Similar to your work schedule, the final report is split into two parts which should contain the following main sections:

Part I – Project assignment (~60% of the project grade)

<u>Section 1. Your FE model(s) set up for the project assignment</u>
Explanation of your model abstraction. Geometry and material parameters, chosen mesh, loads and BC, justification of your modeling approach, list of the assumptions and simplifications made. Presentation of different FE models for the two spring geometries. Mesh convergence establishment, mesh refinement strategy.

<u>Section 2. Numerical results</u>
Relevant numerical results and specific features of interest issued from the FE models, interpretation of the structural response (vs. stress and strain distribution). Comparison of different models used in your project (what do you compare, how and why?).

Section 3. Discussion on the results

Discussing the implication of the modeling assumptions on the numerical results vs. the expected real life behavior. How would you make the model better if time allowed?

Section 4. Conclusions

On the computational results and on the learning.

Section 5. Work distribution within the group and use of AI

List of tasks performed by each student in the group **(compulsory peer-review!)**

Add also a paragraph in this section, following the task list, that explains how generative AI was employed, if it was used.

Section 6. References

List of references used in your project (for the material properties, geometry, closed form expressions, etc.).

Section 7. Figures and tables

Figures and tables related to the project assignment; a maximum 10 figures and/or tables in total, i.e. 5 tables and 5 figures or 9 figures 1 table for example.

Part II – Python implementation (~40 % of the project grade)

Section 9. FE code verification

Maximum 4 pages showing the correct functioning of your FE code: (i) cases and results used for code validation on different levels: single FE and multi-element structure, (ii) convergence study exercise in Salome-meca: different meshes and closed form solution and (iii) comparison to the commercial code results for a given simulation of your project assignment, i.e. the suspension spring. This latter is for bonus points and hence if not performed not penalized.

Note that a serious lack of critical mind in your report induces penalty points on your final project score.

VI - EVALUATION

Your final project result is a function of the final report and your performance during the labworks (including unjustified absence).

By default all group members receive the same grade, but grade dissociation is possible with due justification either on the demand of the group or by the teaching staff.