

Title: FINITE ELEMENT ANALYSIS FOR STRUCTURAL VERIFICATION. FROM MACRO TO MICRO STRUCTURES

Teacher: Morettini, Palmieri

Contact : giulia.morettini@unipg.it, massimiliano.palmieri@unipg.it

Indicative period: April-May 2022

ABSTRACT:

FINITE ELEMENTS ANALYSIS (FEA) plays, historically and for importance, a key role in the structural verification of components in any sector: from large steel structures to small electronic elements. The course aims to provide the basic concepts, theoretical and applicative, on the method of finite element analysis and on the simulation workflow. The focus is placed on the simplest types of analysis: static analysis, modal analysis, and dynamic analysis. The presentation is completed with practical examples.

PROGRAM:

- Finite element method: History and background.
- Theoretical bases of FE approach: definition of finite element, weak form, mass matrix, mass stiffness, methods of reducing system degrees of freedom (Guyan, Craig-Bampton)
- MATLAB implementation of Finite Element procedure for simple plan type problems.
- Presentation of the case studies: from automotive area to electronic components.
- The basic conceptual setting for the FE setting of the Software Ansys Workbench. (brief overview).
- Learn how to model your system using Design Modeler: starting from CAD to a correct FE model.
- Understand the different types of contacts and supports, choose the right type of simulation.
- Learn how to generate the mesh and obtain a mesh independent system.
- How to set the right pre-processing and post-processing setup.
- Interpretation and validation of FE-Solver results.