COMILLAS
UNIVERSIDAD PONTIFICIA
ICAI ICADE CIHS

GUÍA DOCENTE
2023 - 2024

FICHA TÉCNICA DE LA ASIGNATURA

Datos de la asignatura	
Nombre completo	Structural modelling and FEA
Código	DIM-M2S-502
Créditos	3,0 ECTS
Carácter	Obligatoria
Departamento / Área	Departamento de Ingeniería Mecánica

Datos del profesorado	
Profesor	
Nombre	Fernando Martínez Domínguez
Departamento / Área	Departamento de Ingeniería Mecánica
Correo electrónico	fmdominguez@icai.comillas.edu
Profesor	
Nombre	Jaime Espinosa de los Monteros García-Frías
Departamento / Área	Departamento de Ingeniería Mecánica
Correo electrónico	jespinosalosmonteros@icai.comillas.edu
Profesor	
Nombre	Jesús Ángel García Matos
Departamento / Área	Departamento de Ingeniería Mecánica
Correo electrónico	jesus.garcia@iit.comillas.edu

DATOS ESPECÍFICOS DE LA ASIGNATURA

Contextualización de la asignatura

Aportación al perfil profesional de la titulación

After the course students will have a good overview about current simulation capabilities

Prerequisitos

Knowledge of basic courses of algebra and mechanics of materials

Competencias - Objetivos

Competencias

Specialized Knowledge: This course equips students with specialized knowledge and skills in structural modelling and Finite Element

Analysis (FEA), which are valuable assets in various engineering fields.

Design Optimization: Students learn how to optimize the design of structures and components, enhancing their strength and efficiency, and making them well-prepared for engineering challenges.

Safety Expertise: The course covers safety and impact analysis, ensuring that students can prioritize safety in their engineering work across different domains.

Performance Enhancement: Students gain the ability to use FEA to improve the performance of various structures and systems, a skill that can be applied in diverse industries.

Innovation in Technology: The knowledge gained extends beyond the classroom, enabling students to innovate in various technology sectors by designing advanced and efficient systems.

Broad Safety Awareness: The skills acquired can be applied not only in specialized fields but also in everyday applications, contributing to the safety of conventional products and structures.

Competitive Advantage: Completing this course gives students a competitive edge in the job market, as the demand for engineers with expertise in structural modelling and FEA remains high.

The successful completion of the "Structural Modelling and FEA" course within a motorsport, mobility, and safety education enhances students' versatility, competitiveness, and readiness to tackle engineering challenges in various sectors. These skills are valuable assets as they embark on their professional journey.

Resultados de Aprendizaje

This course will be an introduction to the different analyses that can be done in the ANSYS workbench environment and the coupling among them working with solids and Ls-Dyna.
At the end of the course, students have fundamental skills in all the analyses for solids available in Ansys Workbench environment and Explicit Dynamic analysis with Ls-Dyna.

BLOQUES TEMÁTICOS Y CONTENIDOS

Contenidos – Bloques Temáticos

- Introduction to finite elements and workbench environment.
    - Analysis type and workbench workflow.
    - Materials definition.
- Static structural analysis of plane cases. Parameters.
- 3D models.
    - Boundary conditions.
    - Mesh generation.
- Postprocess result.
- Submodelling
- Load cases and load steps.
- Thermal analysis.
- Coupled thermal-structural analysis.
- Static structural analysis of beam and link and shell elements.
- Fundamentals of nonlinear analysis.
    - Non-linear materials and geometric non-linearities

- Fundamentals of contact non-linearities.
    - Types of contact models.
- Modal and harmonic analysis
- Linear buckling analysis
- Rigid Body dynamics
- Transient analysis. Implicit integration
- Ls-Dyna

METODOLOGÍA DOCENTE

Aspectos metodológicos generales de la asignatura

EVALUACIÓN Y CRITERIOS DE CALIFICACIÓN

Test

- Session test
- Final test

Homework in group

- Session cases
- Final project

Calificaciones

Standard evaluation at the end of the term:

- 30% test
    - 40 % session test
    - 60 % final test
- 70 % individual/group works
    - 70 % session cases
    - 30 % final project

Additional evaluation during July (Retake):

- 40% Test
- 60% Individual work

PLAN DE TRABAJO Y CRONOGRAMA

Actividades	Fecha de realización	Fecha de entrega
Sesión Tema Introduction to finite elements		

1	and workbench environment. Analysis type and workbench workflow. Materials definition.		
2	Static structural analysis of plane cases. Parameters.		
3	3D models. Boundary conditions. Mesh generation.		
4	3D models. Postprocess result. Submodelling		
5	Load cases and load steps. Thermal analysis. Coupled thermal-structural analysis .		
6	Static structural analysis of beam and link and shells elements.		
7	Fundamentals of nonlinear analysis. Non-linear materials and geometric non-linearities		
8	Fundamentals of contact non-linearities. Types of contact models.		
9	Modal and harmonic analysis. Linear buckling analysis. Rigid Body dynamics		
10	Transient analysis. Implicit integration		
11	LsDyna1		
12	LsDyna2		
13	LsDyna3		
14	LsDyna4		
15	LsDyna5		

BIBLIOGRAFÍA Y RECURSOS

Bibliografía Básica

- Finite Element Simulations with ANSYS Workbench 18. Huei-Huang Lee. ISBN 978-1630571733.
- Engineering Analysis with ANSYS Workbench 18. Guangming Zhang. ISBN-13: 978-1935673385
- Engineering Analysis with ANSYS Software. Tadeusz Stolarski Y. Nakasone S. Yoshimoto. ISBN: 9780081021644