

课程详述

COURSE SPECIFICATION

以下课程信息可能根据实际授课需要或在课程检讨之后产生变动。如对课程有任何疑问，请联系授课教师。

The course information as follows may be subject to change, either during the session because of unforeseen circumstances, or following review of the course at the end of the session. Queries about the course should be directed to the course instructor.

1.	课程名称 Course Title	计算机仿真设计 Computer Simulation and Design
2.	授课院系 Originating Department	系统设计与智能制造学院 School of System Design and Intelligent Manufacturing
3.	课程编号 Course Code	SDM352
4.	课程学分 Credit Value	3
5.	课程类别 Course Type	专业核心课 Major Core Courses
6.	授课学期 Semester	秋季 Fall
7.	授课语言 Teaching Language	中英双语 English & Chinese
8.	授课教师、所属学系、联系方式 (如属团队授课, 请列明其他授课教师) Instructor(s), Affiliation & Contact (For team teaching, please list all instructors)	熊异, 助理教授 系统设计与智能制造学院 (设计智造学院) XIONG Yi, Assistant Professor School of System Design and Intelligent Manufacturing (SDIM) xiongy3@sustech.edu.cn
9.	实验员/助教、所属学系、联系方式 Tutor/TA(s), Contact	待公布 To be announced
10.	选课人数限额(可不填) Maximum Enrolment (Optional)	待公布 To be announced

11. 授课方式 Delivery Method	讲授 Lectures	习题/辅导/讨论 Tutorials	实验/实习 Lab/Practical	其它(请具体注明) Other (Please specify)	总学时 Total
学时数 Credit Hours	32	0	32	0	64
12. 先修课程、其它学习要求 Pre-requisites or Other Academic Requirements	无 NIL				
13. 后续课程、其它学习规划 Courses for which this course is a pre-requisite	无 NIL				
14. 其它要求修读本课程的学系 Cross-listing Dept.	无 NIL				

教学大纲及教学日历 SYLLABUS

15. 教学目标 Course Objectives

This course aims to equip students with the basic theory and knowledge of finite element analyses (FEA) and computer aided engineering (CAE), and to provide students with the ability to utilize CAE methods and tools for solving structure design problems in the product development. The course consists of three knowledge modules: basics of FEA, applications of FEA, and FEA based structure optimization. More specifically, basics of FEA includes elasticity, and general steps of FEA. Applications of FEA includes utilizing FEA tools for solving various typical problems within the structural design of products. FEA based structure optimization includes the integration of engineering optimization methods with scripting for setting up an automated analysis and design pipeline. The course is conducted with a New Engineering Education pedagogy which equip students with both theoretical and practical knowledge through project-based learning. The main target group of this course is Year 3 students.

本课程主要目标是教授学生有限元与 CAE 的基础理论和基本知识，培养运用计算机仿真设计方法和工具解决产品结构设计问题的能力。课程内容包括三部分：有限元技术基础、有限元技术应用、基于有限元分析的结构优化。其中，有限元技术基础涉及弹性力学、有限元分析基本步骤等。有限元技术应用涉及运用有限元辅助分析软件解决产品结构设计中的各种常见问题。基于有限元分析的结构优化涉及运用工程优化基础方法和编程二次开发实现结构设计的自动化分析和设计。该课程将通过项目引导式的新工科教育方法使学生掌握相关理论和实践知识。本课程以三年级学生为主要对象。

16. 预达学习成果 Learning Outcomes

After passing this course, the student should be able to:

- 1) Understand the general steps of finite element methods.
- 2) Understand the basic finite element formulation techniques.
- 3) Be able to derive equations in finite element methods for 1D, 2D and 3D problems.
- 4) Be able to formulate and solve basic problems in solid mechanics
- 5) Be able to write MATLAB script based on finite element methods.
- 6) Be able to use ABAQUS, a commercial software, to solve basic engineering problems in solid mechanics.

课程结束后，学生将掌握以下技能：

- 1) 理解有限元方法的基本步骤；
- 2) 理解有限元方法的基本原理；
- 3) 能够运用有限元方法推导基本一维、二维、和三维问题；
- 4) 能够运用有限元方法定义和解决固体力学问题；
- 5) 能够使用 MATLAB 编写基本有限元求解代码；
- 6) 能够运用商业求解软件 ABAQUS 解决固体力学中的常见问题。

17. 课程内容及教学日历（如授课语言以英文为主，则课程内容介绍可以用英文；如团队教学或模块教学，教学日历须注明主讲人）

Course Contents (in Parts/Chapters/Sections/Weeks. Please notify name of instructor for course section(s), if this is a team teaching or module course.)

Course Schedule				
Week	Lecture Title	h	Lab Practice	h
1	结构分析 Lecture 1: Structure Analysis	2	Abaqus 概述 Lab 1: Overview of Abaqus	2
2	系统方程式 Lecture 2: System Equations	2	使用 Abaqus 进行有限元分析 Lab 2: FEA using Abaqus	2
3	钢筋结构 Lecture 3: Bar Structure	2	使用 MATLAB 和 Abaqus 分析简单桁架 Lab 3: Analysis of a Simple Truss with MATLAB and Abaqus	2
4	梁结构 Lecture 4: Beam Structure	2	使用 MATLAB 和 Abaqus 分析简单梁 Lab 4: Analysis of a Simple Beam with MATLAB and Abaqus	2
5	虚拟工作原理 Lecture 5: Principle of Virtual Work	2	使用 Abaqus 分析一个简单的框架 Lab 5: Analysis of a Simple Frame with Abaqus	2
6	元素近似 (1) Lecture 6: Element Approximation (1)	2	项目启动 Lab 6: Project Kick-off	2
7	期中回顾 Mid-term Review	2	使用有限元分析固体原件 Lab 7: FEA Using Solid Elements	2
8	元素近似 (2) Lecture 7: Element Approximation (2)	2	使用 CAD 进行有限元建模 (1) Lab 8: FEA with CAD Model (1)	2
9	线弹性 (1) Lecture 8: Linear Elasticity (1)	2	使用 CAD 进行有限元建模 (2) Lab 9: FEA with CAD Model (2)	2
10	线弹性 (2) Lecture 9: Linear Elasticity (2)	2	Abaqus 脚本 (1) Lab 10: Abaqus Scripting (1)	2
11	条形、薄板形和主体 Lecture 10: Bar, Thin-Slab and Body	2	项目中期审查 Lab 11: Project Midterm Review	2
12	梁元素 Lecture 11: Beam Element	2	Abaqus 脚本 (2) Lab 12: Abaqus Scripting (2)	2
13	板元 Lecture 12: Plate-Element	2	结构与优化 (1) Lab 13: Structural Design and Optimization (1)	2
14	问题类型 Lecture 13: Problem Types	2	结构与优化 (2) Lab 14: Structural Design and Optimization (2)	2
15	震动分析 Lecture 14: Vibration Analysis	2	Lab 15: Structural Design and Optimization (3)	2

16	Final Review	2	Lab 16: Project Final Review • Group presentations • Wrapping up of the course	2
----	--------------	---	--	---

18. 教材及其它参考资料 **Textbook and Supplementary Readings**

1. 曾攀. 有限元分析及应用[M]. 清华大学出版社有限公司, 2004. 2. MATLAB 和 Abaqus 有限元分析理论与应用/(澳)Amar Khennane (阿玛尔.哈尼纳尼), 北京:,电子工业出版社:,2020.02

课程评估 ASSESSMENT

19. 评估形式 Type of Assessment	评估时间 Time	占考试总成绩百分比 % of final score	违纪处罚 Penalty	备注 Notes
出勤 Attendance	1-16 周 Week 1-16	10	NIL	
课堂表现 Class Performance				
小测验 Quiz				
课程项目 Projects	1-16 周 Week 1-16	70	NIL	
平时作业 Assignments	1-16 周 Week 1-16	20	NIL	
期中考试 Mid-Term Test				
期末考试 Final Exam				
期末报告 Final Presentation				
其它(可根据需要 改写以上评估方式) Others (The above may be modified as necessary)				

20. 记分方式 **GRADING SYSTEM**

<input checked="" type="checkbox"/> A. 十三级等级制 Letter Grading <input type="checkbox"/> B. 二级记分制(通过/不通过) Pass/Fail Grading

课程审批 REVIEW AND APPROVAL

21. 本课程设置已经过以下责任人/委员会审议通过
This Course has been approved by the following person or committee of authority

--