南方科技大学 SOUTHERN UNIVERSITY OF SCIENCE AND TECHNOLOGY

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**	**有限元理论与工程实践** **Finite Element Theory and Its Engineering Applications**
2.	**授课院系** **Originating Department**	机械与能源工程系 Department of Mechanical and Energy Engineering
3.	**课程编号** **Course Code**	ME314
4.	**课程学分 Credit Value**	3
5.	**课程类别** **Course Type**	专业选修课 Major Elective Courses
6.	**授课学期** **Semester**	春季 Spring
7.	**授课语言** **Teaching Language**	英文 English
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）**	葛锜 副教授 机械与能源工程系 geq@sustech.edu.cn Associate Professor Ge Qi, Dept. Mechanical and Energy Engineering, geq@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）**	

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	42	6			48

12. 先修课程、其它学习要求 Pre-requisites or Other Academic Requirements	线性代数 A（MA107A）、材料力学（MSE305 或者 MAE202）。建议选修 MATLAB 工程应用(ME112) MA107A Linear Algebra A, MSE305 or MAE202 Mechanics of Materials, ME112 Introduction to Matlab (optional)
13. 后续课程、其它学习规划 Courses for which this course is a pre-requisite	
14. 其它要求修读本课程的学系 Cross-listing Dept.	

教学大纲及教学日历 SYLLABUS

15. 教学目标 Course Objectives

通过本课程的学习，将掌握有限元理论的数学方法、简单结构单元(杆和梁)的有限元算法、以及复杂二维/三维问题通过有限元方法的数学表达。在建立了扎实的理论基础之后，通过学习 Abaqus 软件的工程应用案例（一般静态/准静态问题、软材料仿真、结构优化等问题），将熟练掌握 Abaqus 的使用，并能够使用 Abaqus 软件模拟一般性工程问题。

Through this course, student will learn mathematical principles for finite element theory, finite element theory for simple structure elements (bars and beams), and mathematical expressions for finite element theory to solve complex 2D/3D problems. After building the solid theoretical foundation, student will learn how to use Abaqus to solve general engineering problems (general static/quasi-static problems, simulations of soft materials, structural optimizations, etc.) through the process of learning the engineering application examples solved by Abaqus simulations.

16. 预达学习成果 Learning Outcomes

1. 掌握有限元理论的数学方法
2. 掌握简单结构单元(杆和梁)的有限元算法
3. 掌握复杂二维/三维问题通过有限元方法的数学表达
4. 使用 Abaqus 仿真一般静态/准静态问题
5. 使用 Abaqus 仿真软材料问题
6. 使用 Abaqus 实现结构优化

1. Obtain knowledge of the description of mathematical principles for finite element theory.
2. Obtain knowledge of the finite element theory for simple structure elements (bars and beams)
3. Obtain knowledge of mathematical expression for finite element theory to solve complex 2D/3D problems
4. Use Abaqus to simulate general static/quasi-static problems
5. Use Abaqus to simulate soft material problems
6. Use Abaqus to realize structure optimizations

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.)

Topic	Hours	Contents
1. Introduction to Finite Element Method	1	➢ Introduction to Finite Element Method, its history and engineering applications
2. Fundamental Mechanics for Finite element theory	2	➢ Mathematical description of deformable body ➢ Hypotheses of deformable body ➢ Fundamental mechanics equations for 1D problems ➢ Fundamental mechanics equations for 3D problems ➢ Energy Description for deformable body problems
3. Mathematical Principles for Finite Element Theory	3	➢ Analytical solutions for simple problems ➢ Weighted residual method for the approximate solutions of deformable body problems ➢ Virtual work principle, minimum potential energy principle, variation principle for the approximate solutions of deformable body problems
4. Finite Element Theory for bar, beam structures	9	➢ Examples of finding solutions through finite element method ➢ Basic procedures and expressions of finite element analysis ➢ Bar element and its coordinates transformation ➢ Beam element and its coordinate transformation
5. Finite Element Theory for continuum body	9	➢ Element construction for 2D problems ➢ Axisymmetric problems and its element construction ➢ Element construction for 3D problems ➢ General Principle of parametric element
6. Introduction to Abaqus	3	➢ Introduction to Abaqus, its history, and basic CAE operations
7. Static and quasi-static problems	6	➢ Analysis of Axisymmetric problems ➢ Contact problems ➢ Laminate structures
8. Simulation of soft materials by Abaqus	3	➢ Material models of soft materials ➢ Simulation examples of soft material applications
9. Simulation of Heat Transfer Problem	3	➢ Heat Transfer and Thermal Stress Analysis ➢ Heat Transfer in Welding Analysis
10. Introduction to advanced functions of Abaqus	3	➢ Topology Optimization analyses ➢ User Material Subroutine

		➢ UINTER in Abaqus
11. Group discussion and project presentation	6	➢ Group discussion of small projects during the semester ➢ Final project presentation

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

教材（Textbook）：

曾攀，“有限元分析及应用”.

Peter I. Kattan, “MATLAB Guide to Finite Elements”.

参考资料（Supplementary Readings）

王勋成，“有限单元法”.

江丙云等，“ABAQUS”分析之美.

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		5		
课堂表现 Class Performance		10		
小测验 Quiz				
课程项目 Projects				
平时作业 Assignments		25		
期中考试 Mid-Term Test		30		
期末考试 Final Exam				

期末报告 **Final Presentation**		30		
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**

☑ A. 十三级等级制 **Letter Grading** ☐ 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

机械与能源工程系教学委员会