Engineering Design via FEM

Spring 2024, MON/WED, 10:30-11:45
Offline: Creative Learning Building Room 406

Instructor:	Phill-Seung Lee, phillseung@kaist.edu, 010-9105-3694
Teaching Assistants:	
Prerequisites:	Undergraduate Solid Mechanics and Mathematics
References:	*A first course in finite elements*, J. Fish, T. Belytschko, 2007
	Finite element procedures, K.J. Bathe, 1996, *2nd edition*, 2014
Grades:	Homeworks (30%) + Term Project (30%) + Final Exam. (40%)

This course introduces Finite Element Method (FEM), and its applications for mechanical engineering design and analysis at the undergraduate level. The basic principles of FEM are discussed with the help of various mechanics examples with minimal usage of advanced math skills. Typical examples include the coverage of simple elements such as spring, bar and beam elements through the principle of virtual work, and the extension of this concept to the 2D continuum mechanics problems. Students will be exposed to the intensive use of commercial FEM software during practice hours to acquire various modeling and analysis techniques for practical applications.

Schedule (tentative)

Week	Contents	Week	Contents
1	Introduction to FEM Review of solid mechanics	9	Isoparametric finite elements ANSYS session – 4 (Plane strain analysis)
2	Matrix structural analysis (bar & truss system)	10	FE solutions and convergence Plate, shell and 3D solid elements ANSYS session – 5 (Axisymmetric analysis)
3	Principle of virtual work (bar problems)	11	FE solutions and convergence Plate, shell and 3D solid elements ANSYS session – 6 (Plate/shell analysis)
4	Procedure of finite element analysis	12	Dynamics & nonlinear analysis ANSYS session – 7 (3D solid analysis)
5	Introduction to ANSYS ANSYS session – 1 (Truss analysis)	13	Dynamics & nonlinear analysis ANSYS session – 8 (Buckling analysis)
6	Principle of virtual work (2D solid problems) ANSYS session – 2 (Beam analysis)	14	ANSYS session – 9 (Contact analysis) ANSYS session – 10 (Dynamic analysis)
7	Isoparametric finite elements (2D solid elements) ANSYS session – 3 (Plane stress analysis)	15	Term project presentation
8	Midterm examination period (Term project proposal)	16	Final examination